OLYMPUS

User Manual

OLYMPUS Stream

IMAGE ANALYSIS SOFTWARE

Any copyrights relating to this manual shall belong to Olympus Soft Imaging Solutions GmbH.

We at Olympus Soft Imaging Solutions GmbH have tried to make the information contained in this manual as accurate and reliable as possible. Nevertheless, Olympus Soft Imaging Solutions GmbH disclaims any warranty of any kind, whether expressed or implied, as to any matter whatsoever relating to this manual, including without limitation the merchantability or fitness for any particular purpose. Olympus Soft Imaging Solutions GmbH will from time to time revise the software described in this manual and reserves the right to make such changes without obligation to notify the purchaser. In no event shall Olympus Soft Imaging Solutions GmbH be liable for any indirect, special, incidental, or consequential damages arising out of purchase or use of this manual or the information contained herein.

No part of this document may be reproduced or transmitted in any form or by any means, electronic or mechanical for any purpose, without the prior written permission of Olympus Soft Imaging Solutions GmbH.

Adobe and Acrobat are trademarks of Adobe Systems Incorporated and be registered in various countries.

© Olympus Soft Imaging Solutions GmbH

All rights reserved

Version 510_UMA_OlyStream19-Krishna_en_03_09July2013

Olympus Soft Imaging Solutions GmbH, Johann-Krane-Weg 39, D-48149 Münster,
Phone (+49)251/79800-0, Fax: (+49)251/79800-6060

Contents

1. Before you start 6

1.1. Which documentation comes along with your software? 6

Online help for your software 6

1.2. About your software 7

Main features of your software 7

2. User interface 9

2.1. Overview 9

2.2. Layouts 11

2.3. Document group 12

2.4. Tool Windows 13

2.5. Working with documents 14

Saving documents 14

Closing documents 15

Opening documents 15

Activating documents in the document group 16

3. Configuring the system 17

3.1. Overview 17

3.2. Configuring the system 18

4. Acquiring images 21

4.1. Acquiring a single image 21

4.2. HDR images 22

Acquiring an HDR image with an automatically set exposure range 23

Acquiring more HDR images without setting the exposure range anew 24

4.3. Acquiring movies and time stacks 25

Recording a movie 25

Acquiring a time stack 26

4.4. Acquiring a Z-stack 28

Acquiring a Z-stack 28

4.5. Acquiring an EFI image 29

Acquiring an EFI image without a motorized Z-drive 30

4.6. Creating stitched images 34

Acquiring a stitched image without a motorized XY-stage (Manual MIA) 34

Acquiring a stitched image with a motorized XY-stage (XY-Positions / MIA) 38

Acquiring a stitched image with extended depth of focus 41

Automatically acquiring several stitched images 42

Combining individual images into a stitched image 43

5. Processing images 45
5.1. Commenting images 45
5.2. Processing images 45
6. Measuring images 47
6.1. Overview 47
6.2. Measuring images 50
Measuring image objects interactively 50
Outputting various measurement parameters 52
Measuring several images 54
Measuring heights 55
6.3. Measuring welding seams 58
Measuring a throat thickness 58
Performing an asymmetry measurement 62
7. Performing a materials science analysis 65
7.1. Tool window - Materials Solutions 65
Starting an analysis process 66
Selecting the image source 67
Setting the stage path 69
7.2. Chart comparison 76
Performing a chart comparison 77
7.3. Grains Intercept 80
Settings for the intercept analysis 81
Performing an intercept analysis 82
7.4. Grains Planimetric 86
Settings for the planimetric analysis 87
7.5. Layer Thickness 91
Settings for layer thickness measurements 93
Performing an automatic layer thickness measurement 94
Performing a layer thickness measurement with the magic wand (closed layer) 97
Performing a manual layer thickness measurement 101
7.6. Cast Iron Analysis 104
Performing a cast iron analysis (unetched sample) 106
Performing a cast iron analysis (etched sample) 110
7.7. Inclusions Worst Field 113
Performing an inclusions worst field analysis 114
Editing inclusions 116
7.8. Throwing Power 119
Defining throwing power measurements 121
Performing a throwing power measurement 123

7.9. Porosity 128
Thresholds for porosity measurements 130
Counting conditions for porosity measurements 132
Performing a porosity measurement 134
7.10. Phase analysis 138
Performing a phase analysis 141
7.11. Particle Distribution 145
Measuring the particle distribution 147
7.12. Coating Thickness 153
Measuring the coating thickness 154
8. Creating reports 158
8.1. Overview 158
8.2. Working with the report composer 160
8.3. Working with the Olympus MS-Office add-in 164
Creating and editing a new template 165
Editing a report 167
Working with detail zooms 172

1. Before you start

1.1. Which documentation comes along with your software?

The documentation for your software consists of several parts: the installation manual, the online help, and PDF manuals which were installed together with your software.

Where do you find which information?

A quick setup guide describing the software activation is delivered with your software.

On the setup-DVD, several PDF manuals are provided.

- In the installation manual, you can find the system requirements. Additionally, you can find out how to install and configure your software.
- In the user manual, you will find both an introduction to the product and an explanation of the user interface. By using the extensive step-by-step instructions you can quickly learn the most important procedures for using this software.
- The database is explained in its own user manual.

In the online help, you can find detailed help for all elements of your software. An individual help topic is available for every command, every toolbar, every tool window and every dialog box.

New users are advised to use the manual to introduce themselves to the product and to use the online help for more detailed questions at a later date.

Writing convention used in the documentation

In this documentation, the term "your software" will be used for OLYMPUS Stream.

Example images that are automatically installed

During the installation of your software some sample images have been installed, too. These example images might be of help to you when you familiarize yourself with your software.

00018

Online help for your software

In the online help, you can find detailed help for all elements of your software. An individual help topic is available for every command, every toolbar, every tool window and every dialog box.

When you use the help mode, you'll have access to most online topics. As soon as you use the Context help command, you will find yourself in the help mode. In the help mode, a question mark will be attached to the mouse pointer. Then you will be able to call for help on almost all of your software's functions.

Switching to the help mode

There are various ways of switching to the help mode.

- Click the *Context Help* button. You can find this button on the *Standard* toolbar.
- Use the *Help > Context Help* command.
- Use the [Shift + F1] shortcut.

00087

1.2. About your software

Note: Not every software package contains all of the features!

To support the different requirements on the software optimally, a variety of packages are available for your software. The larger software packages contain more features than the smaller packages. For example, the smaller packages contain only restricted database functionality.
Some of the functions described are, therefore, of no relevance to users of smaller packages.

Main features of your software

Acquiring images

You can use your system to acquire high resolution images of a sample in a few steps. Your system is comprised of your software and the hardware, e.g., microscope and camera. During image acquisition, the data from the camera which is mounted on your microscope will be read out and displayed on your computer's monitor.

You can first examine the live-image and adjust it optimally. The live-image will be constantly updated, i.e., when you, for example, move the stage to a different position, the live-image will be changed accordingly. You can switch the live-image on and off and acquire a photo of the parts of the sample that interest you. When you do this, you will create a digital image that you can save and process or analyze with a variety of your software's functions.

Acquiring and viewing multi-dimensional images

A multi-dimensional image is always made up of several frames. These have, for example, been acquired at different times, or in different focus positions. With your software you can, e.g., acquire a time stack or a Z-stack. For optimum viewing of multi-dimensional images, use the separate navigation bar that is shown directly in the image window when a multi-dimensional image is loaded.

Acquiring an EFI image

With your software, you can acquire images which have a practically unlimited depth of focus. These images are called EFI images. EFI is the abbreviation for "Extended Focal Imaging". For the creation of an EFI image, the software determines which of the pixels from the differently focused frames in a Z-stack are the sharpest, and calculates an image that is sharply focused in all areas from them.

Acquiring stitched images

When your system is equipped with a motorized XY-stage: Use the *XY-Positions / MIA* acquisition process to acquire a stitched image of a larger part of the sample. MIA stands for Multiple Image Alignment. During the acquisition, this acquisition process directly combines all of the images that are acquired, into a stitched image, just like a puzzle.

When your system is not equipped with a motorized XY-stage: Use the *Manual MIA* acquisition process and manually move the stage to have the different, adjoining parts of the sample put on display one after the other. By using this acquisition process directly during the acquisition, you combine all of the images that are acquired into a stitched image, just like a puzzle.

Saving documents in a database

You can insert not only images, but also documents which have another file format into a database. That enables you to store all manner of data that belongs together in one location. Search and filter functions make it quick and easy to locate documents.

Images will, by default, be saved in the TIF or VSI file format. When you save an acquired image in TIF format, a lot of important image information will be automatically saved with it, for example, data concerning the camera used, the exposure time, the resolution, the time of creation, and so on. You can later view

this data again whenever you want, simply by opening the image with your software. You do not need to collect this data separately.

A PDF manual for your database is installed together with your software.

Measuring images

You can make various measurements on images, and, e.g., measure the length of a line, the perimeter of an ellipse or an angle in degrees. The measurement objects will be displayed in the image's drawing layer, and can be faded in and out. The measurement results will be shown in a sheet and can be differently sorted by a click of your mouse. You can export measurement results, for example, to the XLS format (for further editing in the MS-Excel application program).

You can measure an image, or several images at the same time, according to different material science analysis processes.
The *Materials Solutions* tool window works similarly to a software wizard. As soon as you've started an analysis process you'll be guided step by step through the measurement.
The following material science analysis processes are available:

- Chart comparison
- Grains Intercept
- Grains Planimetric
- Layer Thickness
- Cast Iron Analysis
- Inclusions Worst Field
- Throwing Power
- Porosity
- Phase Analysis
- Particle Distribution
- Automatic Measurement
- Coating Thickness

Processing images

You can process the acquired images and retroactively optimize the image quality according to your requirements. Numerous filters and functions are available for this purpose, e.g., various smoothing or sharpness filters, and functions to optimize the contrast. As well as this, you can mirror the images and also rotate them through an arbitrary number of degrees.

Analyzing images automatically

With an automatic image analysis, your software searches for areas in an image that have the same intensity, or color. All of the areas that have the same intensity, or color will be assigned to a phase, and evaluated. This makes it possible to automate typical measurement tasks. You can, for example, determine the area ratios of the different phases in an image.

Creating reports

You can document the results of your work in a report. To do this, select the required page templates and images in the *Report Composer* tool window, for example, and generate an MS-Word report.

In case you want to insert images, workbooks or charts from your software into new or existing MS-Word or MS-PowerPoint documents, use a special Olympus add-in for this. With the help of this add-in, you can access all documents and data that you created with your image analysis program from MS-Word or MS-PowerPoint. You can apply different options to all the MS-Word or MS-PowerPoint report's images, detail zooms, for example. It's sufficient for your image analysis program to be open in the background.

Controlling the microscope

You can control your microscope's motorized parts via the software. For example, you can change an objective, load an ND filter, or open and close a shutter, with your software. To make this communication function, the components must not only be motorized, but also have been configured in the software.

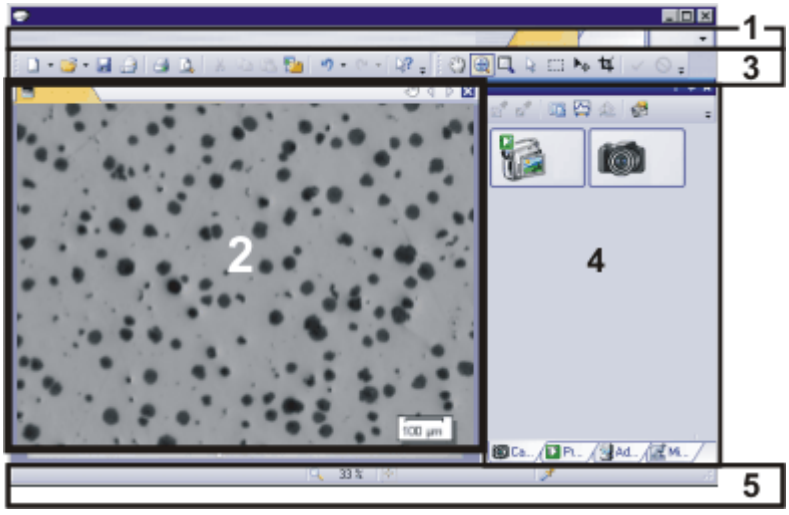
2. User interface

2.1. Overview

The graphical user interface determines your software's appearance. It specifies which menus there are, how the individual functions can be called up, how and where data, e.g. images, is displayed, and much more. Here, the basic elements of the user interface are described.

Note: Your software's user interface can be adapted to suit the requirements of individual users and tasks. You can, e.g., configure the toolbars, create new layouts, or modify the document group in such a way that several images can be displayed at the same time.

Appearance of the user interface



The illustration shows the schematic user interface with its basic elements.

(1) Menu bar
(2) Document group
(3) Toolbars
(4) Tool windows
(5) Status bar

(1) Menu bar

You can call up many commands by using the corresponding menu. Your software's menu bar can be configured to suit your requirements. Use the *Tools > Customization > Start Customize Mode...* command to add menus, modify, or delete them.

You can find more information in the online help.

(2) Document group

The document group contains all loaded documents. These can be of all supported document types.
When you start your software, the document group is empty. While you use your software it gets filled - e.g., when you load or acquire images, or perform various image processing operations to change the source image and create a new one.

(3) Toolbars

Commands you use frequently are linked to a button providing you with quick and easy access to these functions. Please note, that there are many functions which are only accessible via a toolbar, e.g., the drawing functions required for annotating an image. Use the *Tools > Customization > Start Customize Mode...* command to modify a toolbar's appearance to suit your requirements.

(4) Tool windows

Tool windows combine functions into groups. These may be very different functions. For example, in the *Properties* tool window you will find all the information available on the active document.

In contrast to dialog boxes, tool windows remain visible on the user interface as long as they are switched on. That gives you access to the settings in the tool windows at any time.

(5) Status bar

The status bar shows, e.g., a brief description of each function. Simply move the mouse pointer over the command or button for this information.
You can also find additional information in the status bar.

00108

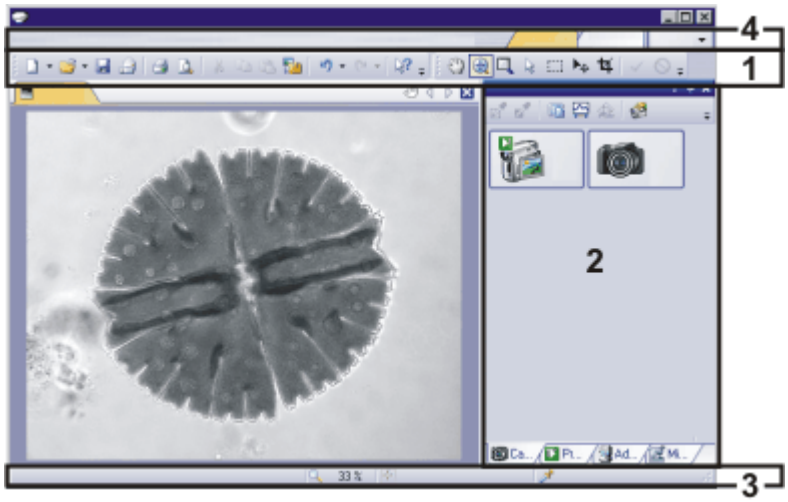
2.2. Layouts

What is a layout?

Your software's user interface is to a great extent configurable, so that it can easily be adapted to meet the requirements of individual users or of different tasks. You can define a so-called "layout" that is suitable for the task on hand. A "layout" is an arrangement of the control elements on your monitor that is optimal for the task on hand. In any layout, only the software functions that are important in respect to this layout will be available.

Example: The *Camera Control* tool window is only of importance when you acquire images. When instead of that, you want to measure images, you don't need that tool window.
That's why the "Acquisition" layout contains the *Camera Control* tool window, whereas in the "Processing" layout it's hidden.

Which elements of the user interface belong to the layout?



The illustration shows you the elements of the user interface that belong to the layout. The layout saves the element's size and position, regardless of whether they have been shown or hidden. When, for example, you have brought the *Windows* toolbar into a layout, it will only be available for this one layout.

(1) Toolbars
(2) Tool windows
(3) Status bar
(4) Menu bar

Switching to a layout

To switch backwards and forwards between different layouts, click on the right-hand side in the menu bar on the name of the layout you want, or use the *View > Layout* command.

Which predefined layouts are there?

For important tasks several layouts have already been defined. The following layouts are available:

- Work with a database ("Database" layout)
- Acquire images ("Acquisition" layout)
- View and process images ("Processing" layout)
- Generate a report ("Reporting" layout)

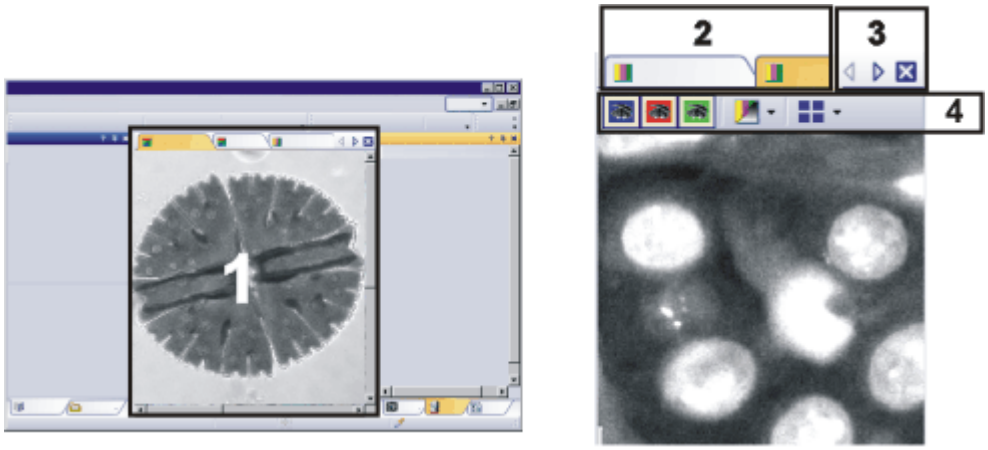
In contrast to your own layouts, predefined layouts can't be deleted. Therefore, you can always restore a predefined layout back to its originally defined form. To do this, select the predefined layout, and use the *View > Layout > Reset Current Layout* command.

00013

2.3. Document group

The document group contains all loaded documents. These can be of all supported document types.

Appearance of the document group



The illustration shows left, a schematic representation of a user interface. On the right, the document group is shown enlarged.

(1) Document group in the user interface
(2) Document bar in the document group
(3) Buttons in the document bar
(4) Navigation bar in the image window

(1) Document group in the user interface

You will find the document group in the middle of the user interface. In it you will find all of the documents that have been loaded, and of course also all of the images that have been acquired. Also the live-image and the images resulting from, e.g., any image processing function, will be displayed there.

(2) Document bar in the document group

The document bar is the document group's header.

For every loaded document, an individual field will be set up in the document group. Click the name of a document in the document bar to have this document displayed in the document group. The name of the active document will be shown in color. Each type of document is identified by its own icon.

(3) Buttons in the document bar

At the top right of the document bar you will see several buttons.

Button with a hand

Click the button with a hand on it to extract the document group from the user interface. In this way you will create a document window that you can freely position or change in size.

If you would like to merge two document groups, click the button with the hand in one of the two document groups. With the left mouse button depressed, drag the document group with all the files loaded in it, onto an existing one.

You can only position document groups as you wish when you are in the expert mode. In standard mode the button with the hand is not available.

Arrow buttons

The arrow buttons located at the top right of the document group are, to begin with, inactive when you start your software. The arrow buttons will only become active when you have loaded so many documents that all of their names can no longer be displayed in the document group. Then you can click one of the two arrows to make the fields with the document names scroll to the left or the right. That will enable you to see the documents that were previously not shown.

Button with a cross

Click the button with a cross to close the active document. If it has not yet been saved, the *Unsaved Documents* dialog box will open. You can then decide whether or not you still need the data.

(4) Navigation bar in the image window

Multi-dimensional images have their own navigation bar directly in the image window. Use this navigation bar to determine how a multi-dimensional image is to be displayed on your monitor, or to change this.

00139

2.4. Tool Windows

What is a tool window?

Tool windows combine functions into groups. These may be very different functions. For example, in the *Properties* tool window you will find all the information available on the active document.

In contrast to dialog boxes, tool windows keep visible on the user interface as long as they are switched on. That gives you access to the settings in the tool windows at any time.

Showing and hiding tool windows

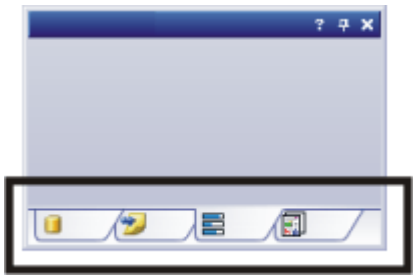
Which tool windows are shown by default depends on the layout you have chosen. You can, at any time, make specific tool windows appear and disappear manually. To do so, use the *View > Tool Windows* command.

Position of the tool windows

The user interface is to a large degree configurable. For this reason, tool windows can be docked, freely positioned, or integrated in document groups.

Docked tool windows

Tool windows can be docked to the left or right of the document window, or below it. To save space, several tool windows may lie on top of each other. They are then arranged as tabs. In this case, activate the required tool window by clicking the title of the corresponding tab below the window.



Freely positioned tool windows

You can only position tool windows as you wish when you are in the expert mode.

You can at any time float a tool window. The tool window then behaves exactly the way a dialog box does. To release a tool window from its docked position, click on its header with your left mouse button. Then, while keeping the left mouse button depressed, drag the tool window to wherever you want it.

Saving the tool window's position

Tool windows and their positions are saved together with the layout and are available at the same position the next time you start your software. Resetting the layout using the *View > Layout > Reset Current Layout* command will have the result that only the tool windows that are defined by default will be displayed.

Buttons in the header

In the header of every tool window, you will find the three buttons *Help*, *Auto Hide* and *Close*.

Click the *Help* button to open the online help for the tool window.
Click the *Auto Hide* button to minimize the tool window.
Click the *Close* button to hide the tool window. You can make it reappear at any time, for example, with the *View > Tool Windows* command.

Context menu of the header

To open a context menu, rightclick a tool window's header. The context menu can contain the *Auto Hide* and *Transparency* commands.

Additionally, the context menu contains a list of all of the tool windows that are available. Every tool window is identified by its own icon. The icons of the currently displayed tool windows will appear clicked. You can recognize this status by the icon's background color.
Use this list to make tool windows appear.

00037

2.5. Working with documents

You can choose from a number of possibilities when you want to open, save, or close documents. As a rule, these documents will be images. In addition, your software supports some other document types. You will find a list of supported documents in the online help.

Saving documents

You should always save important documents immediately following their acquisition. You can recognize documents that have not been saved by the star icon after the document's name.

There are a number of ways in which you can save documents.

1. To save a single document, activate the document in the document group and use the *File > Save As...* command.
2. Use the *Documents* tool window.
   Select the desired document and use the *Save* command in the context menu. For the selection of documents, the standard MS-Windows conventions for multiple selection are valid.
3. Use the *Gallery* tool window.
   Select the desired document and use the *Save* command in the context menu. For the selection of documents, the standard MS-Windows conventions for multiple selection are valid.
4. Save your documents in a database. That enables you to store all manner of data that belongs together in one location. Search and filter functions make it quick and easy to locate saved documents. Detailed information on inserting documents into a database can be found in the online help.

Autosave and close

1. When you exit your software, all data that has not yet been saved will be listed in the *Unsaved Documents* dialog box. This gives you the chance to decide which document you still want to save.
2. With some acquisition processes, the acquired images will be automatically saved after the acquisition has finished.

3. You can also configure your software in such a way that all images are saved automatically after image acquisition. To do so, use the *Acquisition Settings > Saving* dialog box.
   Here, you can also configure your software in such a way that all images are automatically saved in a database after the image acquisition.

Closing documents

There are a number of ways in which you can close documents.

1. Use the *Documents* tool window.
   Select the desired documents and use the *Close* command in the context menu. For the selection of documents, the standard MS-Windows conventions for multiple selection are valid.
2. To close a single document, activate the document in the document group and use the *File > Close* command. Alternatively, you can click the button with the cross . You will find this button on the top right in the document group.
3. Use the *Gallery* tool window.
   Select the desired documents and use the *Close* command in the context menu. For the selection of documents, the standard MS-Windows conventions for multiple selection are valid.

Closing all documents

To close all loaded documents use the *Close All* command. You will find this command in the *File* menu, and in both the *Documents* and the *Gallery* tool window's context menu.

Closing a document immediately

To close a document immediately without a query, close it with the [Shift] key depressed. Data you have not saved will be lost.

Opening documents

There are a number of ways in which you can open or load documents.

1. Use the *File > Open...* command.
2. Use the *File Explorer* tool window.
   To load a single image, double click on the image file in the *File Explorer* tool window.
   To load several images simultaneously, select the images and with the left mouse button depressed, drag them into the document group. For the selection of images the standard MS-Windows conventions for multiple selection are valid.
3. Drag the document you want, directly out of the MS-Windows Explorer, onto your software's document group.
4. To load documents from a database into the document group, use the *Database > Load Documents...* command. You can find more information in the online help.

Generating a test image

If you want to get used to your software, then sometimes any image suffices to try out a function.
Press [Ctrl + Shift + Alt + T] to generate a color test image.
With the [Ctrl + Alt + T] shortcut, you can generate a test image that is made up of 256 gray values.

Activating documents in the document group

There are several ways to activate one of the documents that has been loaded into the document group and thus display it on your monitor.

1. Use the *Documents* tool window. Click the desired document there.
2. Use the *Gallery* tool window. Click the desired document there.
3. Click the title of the desired document in the document group.
4. To open a list with all currently loaded documents, use the [Ctrl + Tab] shortcut. Leftclick the document that you want to have displayed on your monitor.
5. In the *Window* menu, you will find a list of all of the documents that have been loaded. Select the document you want from this list.

Document group and database

Please note that in the *Database* layout the document group will not be shown. Select one of the other layouts, e.g., the *Processing* layout, to have the document group displayed.

00143 08052012

3. Configuring the system

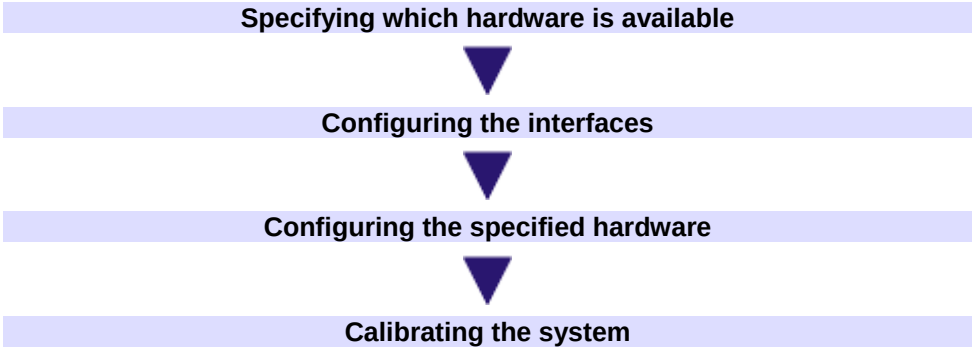
3.1. Overview

Why do you have to configure the system?

After successfully installing your software you will need to first configure your image analysis system, then calibrate it. Only when you have done this will you have made the preparations that are necessary to ensure that you will be able to acquire high quality images that are correctly calibrated. When you work with a motorized microscope, you will also need to configure the existing hardware, to enable the program to control the motorized parts of your microscope.

Process flow of the configuration

To set up your software, the following steps will be necessary:



Specifying which hardware is available

Your software has to know which hardware components your microscope is equipped with. Only these hardware components can be configured and subsequently controlled by the software. In the *Acquire > Devices > Device List* dialog box, you select the hardware components that are available on your microscope.
You can find more information on this dialog box in the online help.

Configuring the interfaces

Use the *Acquire > Devices > Interfaces* command, to configure the interfaces between your microscope or other motorized components, and the PC on which your software runs.
You can find more information on this dialog box in the online help.

Configuring the specified hardware

Usually various different devices, such as a camera, a microscope and/or a stage, will belong to your system. Use the *Acquire > Devices > Device Settings...* dialog box to configure the connected devices so that they can be correctly actuated by your software.
You can find more information on this dialog box in the online help.

Calibrating the system

When all of the hardware components have been registered with your software and have been configured, the functioning of the system is already ensured. However, it's only really easy to work with the system and to acquire top quality images, when you have calibrated your software. The detailed information that helps you to make optimal acquisitions, will then be available.
Your software offers a wizard that will help you while you go through the individual calibration processes. Use the *Acquire > Calibrations...* command to start the software wizard.
You can find more information on this dialog box in the online help.

About the system configuration

When do you have to configure the system?

You will only need to completely configure and calibrate your system anew when you have installed the software on your PC for the first time, and then start it. When you later change the way your microscope is equipped, you will only need to change the configuration of certain hardware components, and possibly also recalibrate them.

Necessary user rights for the system configuration

To be able to configure the system, you have to be logged in to your software with administrator or power user rights. If you have installed the software yourself you will automatically have been assigned Administrator rights.

In contrast, other users who also wish to work with the software will only have user rights as a *Standard User*. With these rights, the system configuration cannot be changed or viewed, i.e., the *Device List* and *Device Settings* dialog boxes cannot be opened.

For this reason, those users who did not themselves install the software, but who are to be allowed to view or change the system configuration, have to be assigned the necessary user rights. Use the *Tools > User Rights...* command to open the *User Rights* dialog box. In it, select the required user, then click the *Properties...* button.

You can find more information on user rights in the online help.

Switching off your operating system's hibernation mode

When you use the MS-Windows Vista operating system: Switch the hibernation mode off.
To do so, click the Start button located at the bottom left of the operating system's task bar.
Use the *Control Panel* command.
Open the *System and Maintenance > Power Options > Change when the computer sleeps* window. Here, you can switch off your PC's hibernation mode.

00159

3.2. Configuring the system

In order to acquire correctly calibrated images, the software requires information about your camera, the objectives and the microscope camera adaptor's magnification. Set up your system with this in mind.

Preconditions

Your software is installed and the camera is connected to your PC. The camera driver is installed in MS-Windows.

Specifying which hardware is available

1. Start your software.

Setting up a new hardware configuration

2. Use the *Acquire > Devices > Device List...* command.
3. Click the *Create New Hardware Configuration* button.
   - The *Create New Hardware Configuration* dialog box will open.
4. Enter a name for the new hardware configuration in the *Name* field. It is a good idea to choose a name combining the microscope and camera names, for example, "BX51_DP25“.
   - Under this name, you can later reload this hardware configuration in the *Device Settings* dialog box.
5. Select the *Copy current hardware configuration* option if you have previously chosen your camera and microscope. Otherwise, choose the *Empty hardware configuration* option.
6. Close the *Create New Hardware Configuration* dialog box with *OK* to return to the *Device List* dialog box.

- You will then find the new hardware configuration entered in the *Configuration* field.
- Once you have completely set up a new hardware configuration, all entries from the *Device List* will be empty. You can now enter a completely new definition for the hardware configuration.

Defining a hardware configuration

Define the new hardware configuration in the *Device List* dialog box. A description of this dialog box can be found in the online help. Begin with the specifications for the camera and the microscope.

7. Select your camera (e.g. "DP25") from the *Camera 1* list.
8. Select your microscope (e.g. "BX51") from the *Frame* list. If your microscope isn't listed, select the *Manual Microscope* entry.
   - Once you have chosen a microscope, the options in the *Device List* dialog box change. For some microscopes there are default settings .

Examples of default settings:

- For the manual microscope BX51, the *Manual Nosepiece* entry from the*Nosepiece* list is preset.
- For the manual stereo microscope SZX10, the *Manual Nosepiece* and *Manual Zoom/Magnification Changer* entries are preset.

9. For some microscopes (such as IX71), you need to choose the port on which your camera is mounted (e.g. *Side (left)*). You find the list to the right of the camera list.
10. All other settings, such as nosepiece, observation filter wheel, shutter and condensor are appropriately preset, independent of your microscope. Check your settings and, if necessary, adjust them to suit your microscope equipment.

Initializing your devices

11. Close the *Device List* dialog box with *OK*.
    - Your hardware configuration will be automatically saved.
    - You can return to the default configuration whenever you want to. To do so, use the *Acquire > Devices > Device Settings...* command. Select the *Default* entry in the *Configuration* list.
    - As soon as you close the *Device List* dialog box, your software will try to set up the connection to the specified devices. You can see whether the devices are able to be successfully controlled in the *Acquire > Devices > Device Settings* device box.

Configuring the specified hardware

1. Use the *Acquire > Devices > Device Settings...* command.
   - In the tree view on the left side, you can find all hardware components that you have chosen in the device list.
2. Select the *Lightpath* entry in the *Sort by* list.

Configuring your camera

3. In the tree view on the left-hand side, expand the *Camera > <camera name>* entry (e.g. "DP25“).
4. Select the *Camera Adapter* entry.
5. Select your camera adapter's magnification on the right-hand side of the *Magnification* list. The magnification is imprinted on your camera adapter. Common values are "1.00" or "0.63".

Configuring the objective nosepiece

6. In the tree structure, either select the *General > Manual Nosepiece* entry (if you have a manual microscope), or *General > Nosepiece <Name of the nosepiece>* (if you have a motorized microscope).

- On the right hand side of the dialog box, the current configuration of the nosepiece will be displayed. When you configure the software for the first time, the fields for the details referring to your objectives will be empty.

7. Choose the objectives which are currently fitted to the nosepiece from the right-hand side of the *Magnification* lists. Start with the smallest magnification and increase up through the higher magnifications. You can read the magnification off of the objective.
8. Choose each corresponding objective from the *Objective Type* list. The type is written on the objective.
   - In the *Description* field, a description of the objective will be suggested. You may change the description of the objective in the *Description* field, if you wish.
9. If the objectives don't use air as their refraction medium, select the immersion medium from the *Refraction Index* list. In this case, you find an appropriate label on the objective.

Configuring the mirror turret

10. Select the *General > <Name of the mirror turret>* entry in the tree view.
11. Make a selection for every position, whether it is occupied or not. For occupied positions, either select a filter or fluorescence cube being used from the *Filter* list, or enter the name of your filter module.
12. Select the *Free* entry for positions that have been purposely left free to keep the light path free of optical elements.
    For example, where the mirror turret is concerned, it is especially important that one position is kept free, in order not to impede the light path for the transmitted light microscopy.

Finishing system configuration

13. Close the *Device Settings* dialog box with *OK*.
    - In certain cases, you receive a message telling you to check the calibrations. You can perform calibration now or later.
14. To have this toolbar displayed, use the *View > Toolbars > Microscope Control* command.
    - The *Microscope Control* toolbar contains buttons with all of your objectives with correct color codes.
    - For stereo microscopes or inverted microscopes, you find the zoom factors in the list to the right of the objectives.

00156

4. Acquiring images

4.1. Acquiring a single image

You can use your software to acquire high resolution images in a very short period of time. For your first acquisition you should carry out these instructions step for step. Then, when you later make other acquisitions, you will notice that for similar types of sample many of the settings you made for the first acquisition can be adopted without change.

1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.
   - You can find the *Microscope Control* (**1**) toolbar at the upper edge of the user interface, right below the menu bar.
     To the right of the document group, you can find the *Camera Control* (**2**) tool window.

Selecting an objective

2. On the *Microscope Control* toolbar, click the button with the objective that you use for the image acquisition.

Switching on the live-image

3. In the *Camera Control* tool window, click the *Live* button.
   - The live-image (**3**) will now be shown in the document group.
4. Go to the required specimen position in the live-image.

Setting the image quality

5. Bring the sample into focus. The *Focus Indicator* toolbar is there for you to use when you are focusing on your sample.
6. Check the color reproduction. If necessary, carry out a white balance.
7. Check the exposure time. You can either use the automatic exposure time function, or enter the exposure time manually.
8. Select the resolution you want.

Acquiring and saving an image

9. In the *Camera Control* tool window, click the *Snap* button.
   - The acquired image will be shown in the document group.
10. Use the *File > Save As...* command to save the image. Use the recommended TIF file format.

00027

4.2. HDR images

What are HDR images?

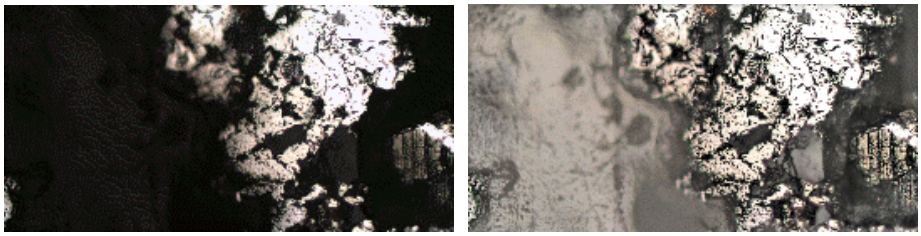
Under the microscope, certain samples (with very reflective metal surfaces, for example) can have such strong differences in brightness that it is impossible to find an exposure time that is suitable for all areas of the sample.

For such samples, an HDR image acquisition is recommended. HDR stands for High Dynamic Range. Dynamic range relates to the capacity of cameras, or image processing software, to display both the bright and the dark segments of an image well.

Before acquiring an HDR image, the necessary exposure range needs to be determined for the current sample. The exposure range is made up of a minimum and maximum exposure time as well as several exposure times between them. Several individual images are then taken of the sample with differing exposure times, so that no image segment is left over or underexposed.

Your software then detects the best exposed pixels in each acquired individual image and merges them into one new image. Under correctly defined acquisition conditions, the HDR image will no longer contain any under or overexposed image segments.

Just like with images acquired with Extended Focal Imaging (EFI image), an HDR image is a rendered image containing information from several images.



Here, you can see an image acquired of a very reflective metal surface. Example 1 shows an image which was not acquired using HDR. The reflective segments of the surface are correctly exposed, whereas other segments are completely underexposed. Example 2 shows an image which was acquired using HDR. Without overexposing the reflective segments of the surface, now the structures in the dark image segments, which were not recognizable before, are visible.

Determining the exposure range

A recently determined exposure range will continue to be used for all HDR images until you let your software determine the exposure range anew. It is irrelevant whether the exposure range had been determined automatically or manually.

If you are acquiring several images of the same or similar parts of a sample, you don't need to determine the exposure range each time. If you change the sample or adjust settings on the microscope, it is recommended to determine the exposure range anew (either automatically or manually).

HDR images and acquisition processes

You can also insert an HDR image acquisition into an acquisition process, such as during the acquisition of a time-lapse image or a Z-stack. The *Process Manager* tool window informs you about the status of the HDR image acquisition. If the *Activate HDR* check box is selected in the *Camera control* tool window, the *Process Manager* tool window shows the *Active* entry in the *HDR* field. If the check box is deselected, the *Process Manager* tool window shows the entry *Off* in the *HDR* field.

HDR images and movie recording

It is not possible to record movies with HDR. Because of this, the *Activate HDR* check box is ignored while the *Movie recording* check box is selected.

07510 04072011

Acquiring an HDR image with an automatically set exposure range

With this procedure, your software automatically determines the exposure range. To do so, your camera automatically acquires a set of images with various exposure times and measures the amount of over and underexposed pixels. The exposure time continues to change until the amount of over and underexposed pixels is within defined limits. At this point, the exposure range has been defined. How much the exposure time is adjusted by is determined by your software with regards to the minimum and maximum exposure time.

Preparations

1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.
2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the acquisition of the HDR image.
3. Switch to the live mode, and select the optimal settings for your acquisition, in the *Camera Control* tool window. Carry out a white balance. Then select an exposure time meaning that no part of the sample is overexposed.
   - The automatic exposure time detection uses this value as a basis and raises the exposure time so as to correctly light even the dark parts of the sample.
4. Search for the part of the sample which you want to acquire an HDR image of. This should be a position which has such significant differences in brightness that not all segments can be shown with optimal lighting.
5. Finish the live mode.

Acquiring an HDR image

6. In the *Camera Control* tool window, select the *Activate HDR* check box.
   - In the upper part of the tool window, the *Snap* button changes to the *HDR* button.
7. In the *Determine exposure range* group, click the *Automatic* button to have the exposure range determined automatically.
   - The necessary exposure range will now be determined. To do so, the camera automatically acquires several images which only differ in exposure time. This acquisition occurs in the background, which means that the images are not shown in the document group. The exposure range determined in this way will continue to be used for all HDR images until you let your software determine the exposure range anew.
   - Determining the exposure range automatically takes about 30 seconds. Pay attention to the progress bar located in the status bar. When all elements in the tool window are active again, the process has finished. In the *Total time* field, you can now see how long is needed for the HDR image acquisition.
   - If, in the *Acquisition Settings > Acquisition > HDR* dialog box, the *Automatic HDR preview* check box is selected, the HDR image will be acquired and shown automatically, once the exposure range has been set.
8. If the HDR image has not been acquired automatically, click the *HDR* button in the *Camera Control* tool window to start the image acquisition.
   - The image acquisition will begin. Pay attention to the progress bar located in the status bar

. It shows how long the acquisition has taken and the total acquisition time. The

progress bar contains the *Cancel* button, which you can use to stop the current image acquisition.

- After the acquisition has been completed the HDR image will be shown in the document group.

9. Check the image. If you want to change the settings (to use a different algorithm for the output rendering, for example), open the *Acquisition Settings* dialog box. Select the *Acquisition > HDR* option in the tree view.
   - You can find more information on this topic in the online help.
10. If you don't want to change any settings, use the *File > Save As...* command to save the image. Use the recommended TIF or VSI file format.
    - These are the only formats which also save all the image information including the HDR entries together with the image. This means that you can always see whether or not an image was acquired using HDR. Open the *Properties* tool window, and look at the data in the *Camera* group.

Acquiring more HDR images without setting the exposure range anew

If you have just acquired HDR images of the same or a similar sample, as a rule, it is not necessary to determine the dynamic range anew. In this case, you have already completed the preparations for acquisition (such as carrying out a white balance) and set the HDR image acquisition settings correctly (such as choosing the optimal algorithm used for output rendering) anyway.

In such circumstances, acquiring an HDR image is especially easy. Do the following:

1. In the *Camera Control* tool window, select the *Activate HDR* check box.
2. Click the *HDR* button in the *Camera Control* tool window to start the image acquisition.
   - The image acquisition will begin. After the acquisition has been completed the HDR image will be shown in the document group.
3. Check the image before saving it.
   - This step can be left out if your software is configured to import images into a database directly after acquisition.

07500 15072011

4.3. Acquiring movies and time stacks

With your software you can acquire movies and time stacks.

Recording a movie

You can use your software to record a movie. When you do this, your camera will acquire as many images as it can within an arbitrary period of time. The movie will be saved as a file in the AVI format. You can use your software to play it back.

1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.

Selecting an objective

2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the movie acquisition.

Selecting the storage location

3. In the *Camera Control* tool window's toolbar, click the *Acquisition Settings* button.
   - The *Acquisition Settings* dialog box opens.
4. Select the *Saving > Movie* entry in the tree structure.
5. You have to decide how a movie is to be saved after the acquisition. Select the*Filesystem* entry in the *Automatic save > Destination* list to automatically save the movies you have acquired.
   - The *Path* field located in the *Directory* group shows the directory that will currently be used when your movies are automatically saved.
6. Click the [...] button next to the *Path* field to alter the directory.

Selecting the compression method

   - The AVI file format is preset in the *File type* list. This is a fixed setting that cannot be changed.
7. Click the *Options...* button when you want to compress the AVI file in order to reduce the movie's file size.
8. From the *Compression* list, select the *M-JPEG* entry and confirm with *OK*.

Please note: Compressing the movie is only possible if the selected compression method (codec) has already been installed on your PC. If the compression method has not been installed the AVI file will be saved uncompressed.
The selected compression method must also be available on the PC that is used for playing back the AVI. Otherwise the quality of the AVI may be considerably worse when the AVI is played back.

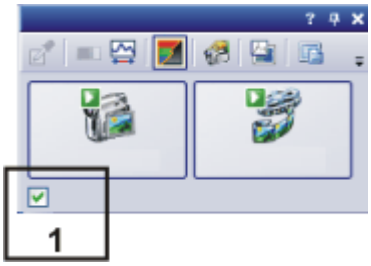
9. Close the *Acquisition Settings* dialog box with *OK*.

Setting the image quality

10. Switch to the live mode, and select the optimal settings for movie recording in the *Camera Control* tool window. Pay special attention to setting the correct exposure time.
    - This exposure time will not be changed during the movie recording.
11. Find the segment of the sample that interests you and focus on it.

Switching to the "Movie recording" mode

12. Select the *Movie recording* check box (**1**). The check box can be found below the *Live* button in the *Camera Control* tool window.



- The *Snap* button will be replaced by the *Movie* button.

Starting movie recording

13. Click the *Movie* button to start the movie recording.
    - The live-image will be shown and the recording of the movie will start immediately.
    - In the status bar a progress indicator is displayed. At the left of the slash the number of already acquired images will be indicated. At the right of the slash an estimation of the maximum possible number of images will be shown. This number depends on your camera's image size and cannot exceed 2GB.

    - This icon on the *Movie* button indicates that a movie is being recorded at the moment.

Stopping movie recording

14. Click the *Movie* button again to end the movie recording.
    - The first image of the movie will be displayed.
    - The navigation bar for time stacks will be shown in the document group. Use this navigation bar to play the movie.
    - The software will remain in the "Movie recording" mode until you clear the *Movie recording* check box once more.

Acquiring a time stack

In a time stack all frames have been acquired at different points of time. With a time stack you can document the way the position on the sample changes with time. To begin with, for the acquisition of a time stack make the same settings in the *Camera Control* tool window as you do for the acquisition of a snapshot. Additionally, in the *Process Manager* tool window, you have to define the time sequence in which the images are to be acquired.

Task

You want to acquire a time stack over a period of 10 seconds. One image is to be acquired every second.

1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.

Selecting an objective

2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the image acquisition.

Setting the image quality

3. Switch to the live mode, and select the optimal settings for your acquisition, in the *Camera Control* tool window. Pay special attention to setting the correct exposure time. This exposure time will be used for all of the frames in the time stack.
4. Choose the resolution you want for the time stack's frames, from the *Resolution > Snap/Process* list.
5. Find the segment of the sample that interests you and focus on it.

Selecting the acquisition process

6. Activate the *Process Manager* tool window.
7. Select the *Automatic Processes* option.
8. Click the *Time Lapse* button.
   - The button will appear clicked. You can recognize this status by the button's colored background.
   - The [*t*] group will be automatically displayed in the tool window.
9. Should another acquisition process be active, e.g., *Z-Stack*, click the button to switch off the acquisition process.
   - The group with the various acquisition processes should now look like this:

Selecting the acquisition parameters

10. Clear the check boxes *Start delay* and *As fast as possible*.
11. Specify the time that the complete acquisition is to take, e.g., 10 seconds. Enter the value "00000:00:10" (for 10 seconds) in the *Recording time* field. You can directly edit every number in the field. To do so, simply click in front of the number you want to edit.
12. Click the button with the lock located to the right of the field, to specify that the acquisition time is no longer to be changed.
13. Specify how many frames are to be acquired.
    Enter e.g., 10 in the *Cycles* field.
    - The *Interval* field will be updated. It shows you the time that will elapse between two consecutive frames.

Acquiring a time stack

14. Click the *Start* button.
    - The acquisition of the time stack will start immediately.
    - The *Start Process* button changes into the *Pause* button. A click on this button will interrupt the acquisition process.
    - The *Stop* button will become active. A click on this button will stop the acquisition process. The images of the time stack acquired until this moment will be preserved.
    - At the bottom left, in the status bar, the progress bar will appear. It informs you about the number of images that are still to be acquired.
    - The acquisition has been completed when you can once more see the *Start* button in the *Process Manager* tool window, and the progress bar has been faded out.
    - You will see the time stack you've acquired in the image window. Use the navigation bar located in the image window to view the time stack. You can find more information on the navigation bar in the online help.
    - The time stack that has been acquired will be automatically saved. The storage directory is shown in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.

Note: When other programs are running on your PC, for instance a virus scanning program, it can interfere with the performance when a time stack is being acquired.

00304

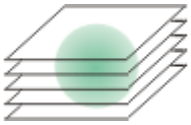
4.4. Acquiring a Z-stack

A Z-stack contains frames acquired at different focus positions. That is to say, the microscope stage was located in a different Z-position for the acquisition of each frame.

Note: You can only use the *Z-Stack* acquisition process when your stage is equipped with a motorized Z-drive.

Acquiring a Z-stack

Example: You want to acquire a Z-stack. The sample is approximately 50 µm thick. The Z-distance between two frames is to be 2 µm.



1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.

Selecting an objective

2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the image acquisition.

Setting the image quality

3. Switch to the live mode, and select the optimal settings for your acquisition, in the *Camera Control* tool window. Pay special attention to setting the correct exposure time. This exposure time will be used for all of the frames in the Z-stack.
4. Search out the required position in the sample.

Selecting the acquisition process

5. Activate the *Process Manager* tool window.
6. Select the *Automatic Processes* option.
7. Click the *Z-Stack* button.
   - The button will appear clicked. You can recognize this status by the button's colored background.
   - The [*Z*] group will be automatically displayed in the tool window.

Selecting the acquisition parameters

8. Select the *Range* entry in the *Define* list.
9. Enter the Z-range you want in the *Range* field. In this example, enter a little more than the sample's thickness (= 50 µm), e.g., the value 60.
10. In the *Step Size* field, enter the required Z-distance, e.g., the value 2, for a Z-distance of 2 µm. The value should roughly correspond to your objective's depth of focus.
    - In the *Z-Slices* field you will then be shown how many frames are to be acquired. In this example 31 frames will be acquired.
9. Find the segment of the sample that interests you and focus on it. To do this, use the arrow buttons in the [*Z*] group. The buttons with a double arrow move the stage in larger steps.

Acquiring an image

10. Click the *Start* button.
    - Your software now moves the Z-drive of the microscope stage to the start position. The starting positions lies half of the Z-range deeper than the stage's current Z-position.

- The acquisition of the Z-stack will begin as soon as the starting position has been reached. The microscope stage moves upwards step by step and acquires an image at each new Z-position.
- The acquisition has been completed when you can once more see the *Start* button in the *Process Manager* tool window, and the progress bar has been faded out.
- You can see the acquired Z-stack in the image window. Use the navigation bar located in the image window to view the Z-stack. You can find more information on the navigation bar in the online help.
- The Z-stack that has been acquired will be automatically saved. You can set the storage directory in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.

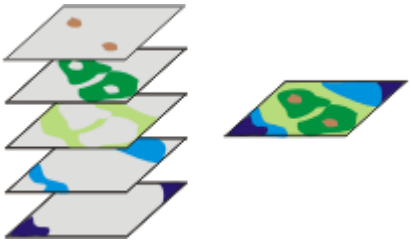
Note: When other programs are running in the background on your PC, for instance a virus scanning program, it can interfere with the performance when a Z-stack is being acquired.

00367

4.5. Acquiring an EFI image

What is EFI?

EFI is the abbreviation for Extended Focal Imaging. By using the "EFI" acquisition process you can acquire images with your microscope which have practically unlimited depth of focus. To do this, EFI uses a series of differently focused separate images (Focus series) to calculate a resulting image (EFI image), that is focused in all of its parts.



At the left hand-side, the illustration shows a number of frames that were acquired at different Z-positions. In each of these frames there are only a few image segments that are displayed sharply focused. These segments are shown in color. These sharply focused image segments will be assembled into the EFI image (right).

Creating an EFI image

Your software offers you several ways of creating an EFI-image.

- Acquiring an EFI image without a motorized Z-drive
- Acquiring an EFI image with a motorized Z-drive

Acquiring an EFI image without a motorized Z-drive

Task

You have a thick section in the transmitted light mode, or a sample with a very rough surface in the reflected light mode, e.g., with holes, grooves, bumps peaks or slanting planes. In the image it's only possible to bring one layer of the section, or only part of the surface, sharply into focus, higher-lying or deeper-lying areas are outside the depth of focus range. Acquire a Z-stack through the complete thickness or height of the sample, and have the EFI image calculated for you.

In this case, you can use the manual *Instant EFI* acquisition process to acquire a sharply focused image of all of the sample.

Note: You can use the *Instant EFI* acquisition process with every microscope. When your microscope stage is equipped with a motorized Z-drive, use the *Z-Stack* acquisition process to acquire an EFI image.

Selecting the acquisition process

1. Use the *View > Tool Windows > Process Manager* command to make the *Process Manager* tool window appear.
2. Select the *Manual Processes* option.
3. Click the *Instant EFI* button.
   - The button will appear clicked. You can recognize this status by the button's colored background.
   - The *Instant EFI* group will be automatically displayed in the tool window.

Setting acquisition parameters

4. From the *Algorithm* list, select the *Reflected light* entry, when you use your light, or stereo microscope in the reflected light mode.
5. If you work with a stereo microscope, select the *Automatic frame alignment* check box.
   If you don't work with a stereo microscope, clear the *Automatic frame alignment* check box.

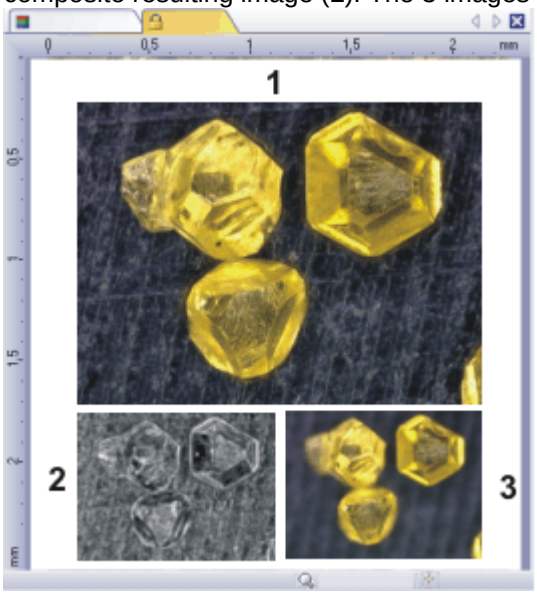
Preparing an EFI acquisition

6. Use the *View > Tool Windows > Camera Control* command to make the *Camera Control* tool window appear.
7. In the *Camera Control* tool window, click the *Live* button.
8. Move the microscope focus to the Z-position where either the lowest or the highest place on the sample is only just no longer sharply focused. Use the live-mode for a visual control.
9. Check the exposure time, and correct it if necessary. When the *Instant EFI* acquisition process has been started, the exposure time will be kept constant during the whole of the acquisition.

Acquiring an EFI image

10. In the *Process Manager* tool window, click the *Start* button.

- The live-image in the document group will divide itself into 3 images. On the bottom right, you'll still see the live-image (**3**). On the bottom left, you'll see the sharpness map (**2**). The large image above them is the composite resulting image (**1**). The 3 images will be continually updated.



11. Use your microscope's Z-drive to move your stage slowly through the height range of the sample's surface.
    - Your software will acquire images at the various focal planes, then it will set them together. While this is being done, the camera will acquire the images as quickly as possible. The sharpness value of individual pixels will be calculated for every image. If the sharpness values are higher than in the previous images, the pixels in the composite EFI image will be adopted. The EFI image contains the pixels with the highest sharpness values from all of the images acquired up till then.
    - The sharpness map at the bottom left will show you which image areas will be sharply reproduced in the EFI image. The brighter a pixel is in the sharpness map, the higher is its sharpness value in the EFI image.
    - Once the acquisition process has been started, the sharpness map should only be bright at the deepest, or highest parts of the sample, the rest of the map is dark.
12. Focus on the sample slowly once through all the focal planes.
    After each change of the focus position, wait until you see that further areas become brighter in the sharpness map.
    - As the process continues, more and more areas in the sharpness map should become brighter. At the same time the EFI image will also get better and better.
13. Check the EFI image and the sharpness map. Are all areas of the image now sharp? Are there any areas in the sharpness map that are still dark?
    Focus on these areas and have additional images calculated into the EFI image. Continue acquiring additional images until the whole sample has been sharply reproduced.
14. In the *Process Manager* tool window, click the *Stop* button.
    - The resulting image is not a Z-stack, but a standard image.
    - The EFI image will be automatically saved. You can set the storage directory in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.

15. In the *Camera Control* tool window, click the *Live* button again.

Acquiring an EFI image with a motorized Z-drive

Task

You have a thick section in the transmitted light mode, or a sample with a very rough surface in the reflected light mode, e.g., with holes, grooves, bumps peaks or slanting planes. In the image it's only possible to bring one layer of the section, or only part of the surface, sharply into focus, higher-lying or deeper-lying areas are outside the depth of focus range. Acquire a Z-stack through the complete thickness or height of the sample, and have the EFI image calculated for you.

You can use the automatic *Z-stack* acquisition process to acquire a sharply focused image of all of the sample.

Prerequisite: You can only use the *Z-Stack* acquisition process when your stage is equipped with a motorized Z-drive.

Setting the EFI parameters

1. Activate the *Process Manager* tool window.
2. To open the *Acquisition Settings* dialog box, click the *Acquisition Settings* button in the tool window's toolbar.
3. Select the *Acquisition > Automatic EFI* entry in the tree view.
4. In the *Algorithm* list, select the *Transmitted light (exp)* entry, if you're working in the transmitted light mode, and the *Reflected light* entry, if you're working in the reflected light mode.
5. Select the *Automatic frame alignment* check box when you're working with a stereo microscope and acquiring the sample at a viewing angle. Otherwise, clear this check box.
6. Close the *Acquisition Settings* dialog box with *OK*.

Preparing for the acquisition of a Z-stack

7. Carry out all the microscope settings.
8. On the *Microscope Control* toolbar, click the button corresponding to the objective you've set.
9. Activate the *Camera Control* tool window.
10. Switch to the live mode.
11. Optimize the exposure time. The exposure time will be kept constant during the acquisition of the Z-stack.
12. Click the *Autofocus* button in the *Camera Control* tool window's toolbar to focus.

Setting the Z-stack parameters

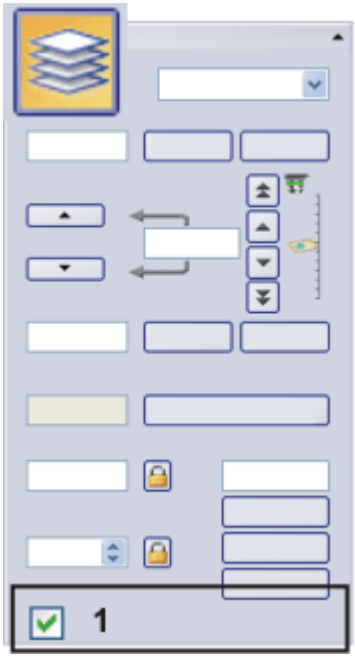
13. Activate the *Process Manager* tool window.
14. Select the *Z-Stack* acquisition process.
15. Select the *Top and bottom* entry in the *Define* list.
16. Use the arrow buttons in the [*Z*] group to move your stage to the Z-position at which the lowest-lying position on the sample is sharply focused. The arrow buttons move the stage either by steps of 2 µm or of 20 µm.
    - The stage's current position will be shown to you in the*Pos.* field.
17. Click the top *Set* button to define the starting position for the Z-stack acquisition.
    - The current Z-position will be adopted in the *End* field.
18. Use the arrow buttons in the [*Z*] group to move your stage to the Z-position at which the highest-lying position on the sample is sharply focused.
19. Click the bottom *Set* button to define the position at which the Z-stack acquisition is to end.

- The current Z-position will be adopted in the *End* field.

20. In the *Step Size* field, enter the distance between two frames in the Z-stack. This Z-distance should be small enough to ensure that no positions on the sample between two images remain blurred. The higher your objective's Numerical Aperture is, the smaller the Z-spacing should be.
21. Use the [Enter] key to confirm the Z-distance that you've set.
    - The number of images in the stack will be automatically calculated on the basis of the Start and End values, and the Z-distance.

Starting an EFI acquisition

22. Select the *Extended Focal Imaging* (**1**) check box. You can find the check box at the bottom of the [*Z*] group located in the *Process Manager* tool window.



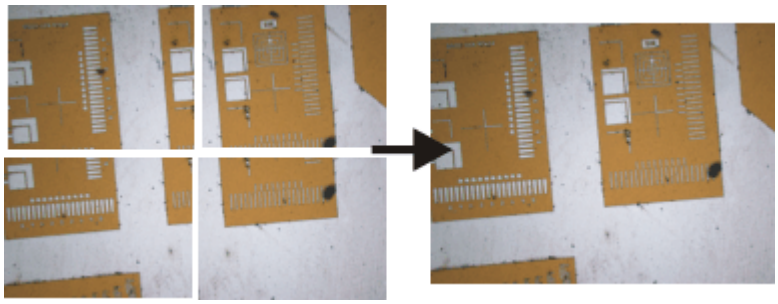
23. Finish the live mode.
24. Click the *Start* button.
    - The EFI acquisition begins immediately.
    - The acquisition will begin. After the acquisition has been completed the EFI image will be shown in the document group. This image was calculated from the variously focused separate images.

00372 05072012

4.6. Creating stitched images

What is a stitched image?

If you acquire a stitched image, move the stage in a way that different, adjoining parts of the sample are shown. All of the images that are acquired are combined, just like a puzzle, into a stitched image. The stitched image will display a large part of the sample in a higher X/Y-resolution than would be possible with a simple snapshot.



The illustration shows left, four separate images. On the right you see the stitched image made up from the four images.

Creating a stitched image

Your software offers you several ways of creating a stitched image.

- Acquiring a stitched image without a motorized XY-stage (Manual MIA)
- Acquiring a stitched image with a motorized XY-stage (XY-Positions / MIA)
- Acquiring a stitched image with extended depth of focus
- Automatically acquiring several stitched images
- Combining individual images into a stitched image

Materials science analysis processes on stitched images

The *Materials Solutions* tool window offers several materials science analysis processes. Most of these processes can also use stitched images an input, provided that you are working with a motorized XY-microscope stage. The acquisition of these stitched images are defined in the *Stage path settings* step within the materials science analysis process. In this case, you won't need the acquisition processes described in this topic.

Acquiring a stitched image without a motorized XY-stage (Manual MIA)

Task

You want to acquire an image of a large sample area. Use the *Manual MIA* acquisition process to acquire several individual images of adjoining positions on the sample, and to have them combined into a stitched image. MIA stands for Multiple Image Alignment.

Prerequisite

The camera is aligned parallel to the XY-stage. The angle between camera and stage should be smaller than 1°.

1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.

Selecting an objective

2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the acquisition of the stitched image.

Setting the image quality

3. Switch to the live mode, and select the optimal settings for your acquisition, in the *Camera Control* tool window. Pay special attention to setting the correct exposure time. This exposure time will be used for all of the stitched image's individual images.

4. Find the position on the sample at which you want to start acquiring the stitched image.
5. Finish the live mode.

Selecting the acquisition process

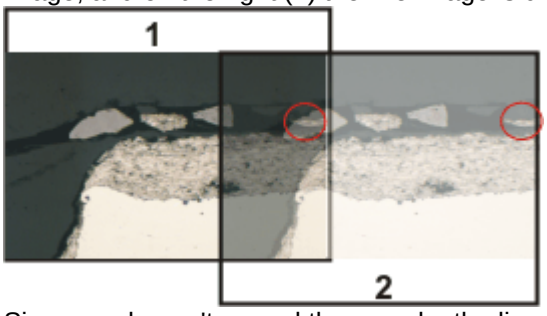
6. Activate the *Process Manager* tool window.
7. Select the *Manual Processes* option.
8. Click the *Manual MIA* button.
   - The button will appear clicked. You can recognize this status by the button's colored background.
   - The *Manual MIA* group will be automatically displayed in the tool window.
   -
     Should the *Instant EFI* acquisition process have been active, it will be automatically switched off. You can, however, use images with extended depth of focus for the stitched image. To do this, before you acquire each of the individual images, click the *Instant EFI* button located in the *Manual MIA* group.

Selecting the acquisition parameters

9. Make quite certain that the *Auto Align* button appears clicked. It should then look like this:
   .
   - Then your software will search for the same image structures in neighboring individual images. The stitched image will be put together in such a way that image areas that are the same will be superimposed.

Acquiring a stitched image

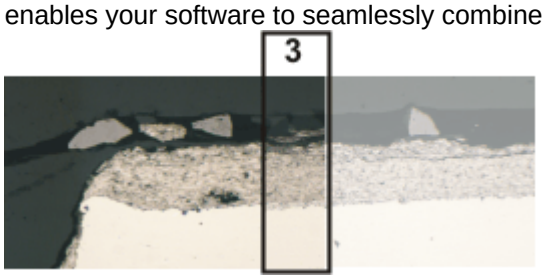
10. Click the *Start* button.
    - Your software switches into the live mode.
11. Bring the sample into focus.
12. Click on one of the arrow buttons to set the side of the current image at which the next image is to be arranged. For example, click this button if the next image is to be laid to the right of the current image.
    - Your system now acquires an image at the current position on the sample. In the image window you now see on the left (**1**) the acquired image, and on the right (**2**) the live-image is displayed.

      Since you haven't moved the sample, the live-image still shows the current sample position, too, which means that you now see the current image twice.
      The two images overlap. Since the live-image is shown transparent, you see both images in the overlap area simultaneously.
13. Make a note of a significant structure on the live-image's right border. You will find the same sample structure in the overlap area. On the illustration, a significant structure has been indicated by a circle.

14. Now move the stage very slowly to make the structure on the live-image move to the left. Keep moving the stage until the image structures in the overlap area lie as exactly over each other as possible. The image structures need not lie precisely over each other, since your software will match the individual images with each other.

    - In the overlap area (**3**), the same image segments are shown now. This enables your software to seamlessly combine the two images.

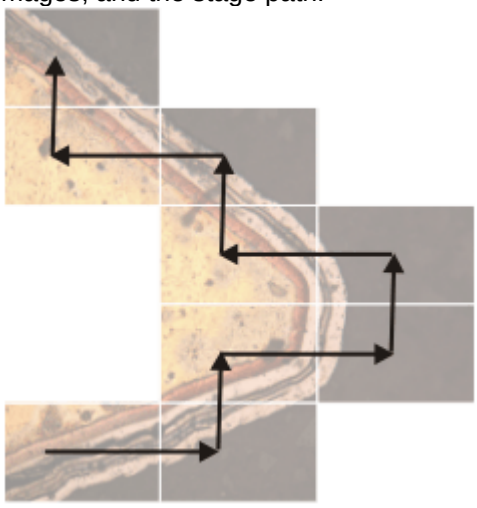


    - You can reverse the direction in which your stage moves, in the *Device Settings > Stage* dialog box. Depending on how you can best orient yourself, the live-image will then move to the left or to the right, when you move your stage to the right.

15. Check whether both images have been correctly combined. Otherwise, you can undo the last step by using the *Undo last frame* button. You can then move the stage again, and match the structures better.

    - During the acquisition, you can change the current stitched image's zoom factor, e.g., to see certain parts in the overlap area better. You will find an overview on the possibilities of changing an image's zoom factor in the online help.

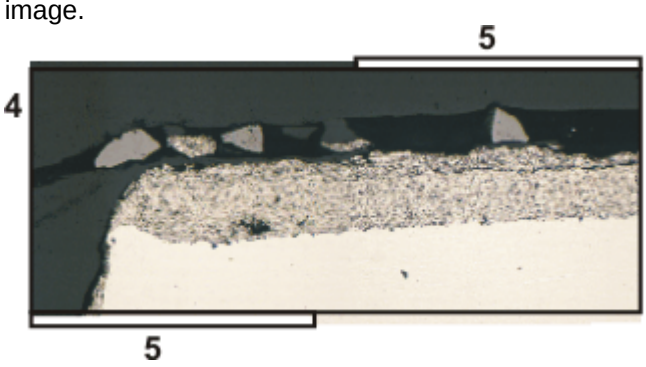
16. Define your way through the sample, with the arrow buttons, and follow that with the stage.
    In this manner, you can display a sample in any form you like in the stitched image. The illustration shows a stitched image that is made up of 9 individual images, and the stage path.



17. Click the *Stop* button when you want to end the acquisition of the stitched image.

    - You see the completed stitched image (**4**) in the image window. Since the individual images can lie a little askew of each other, the stitched image isn't as a rule, rectangular, but contains empty areas on its borders (**5**). These areas will, as a rule, be cut off in the stitched

image.



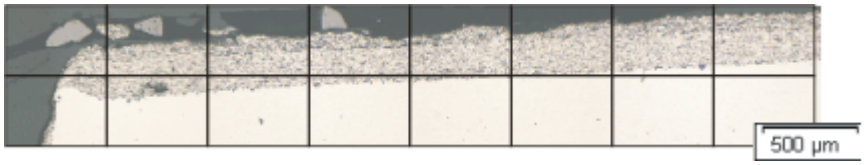
- The stitched image will, by default, be automatically saved. The storage directory is shown in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.

Properties of the stitched image

- By default, in the overlap area, the intensity values of two adjoining individual images will be matched with each other to make the image's overall impression homogeneous.
- Stitched images are calibrated. This means that you can measure distances and objects on a stitched image.

Acquiring a stitched image with a motorized XY-stage (XY-Positions / MIA)

Task



You want to acquire an image of a large sample area. Use the automatic *XY-Positions / MIA* acquisition process to scan a rectangular area of the sample and to have adjoining images combined into one stitched image. MIA stands for Multiple Image Alignment.

You can only use the *XY-Positions/ MIA* acquisition process if your microscope is equipped with a motorized XY-stage.

Preconditions

- The stage has been set up and initialized, i.e. its stage limits have been defined.
- The camera is aligned parallel to the XY-stage. The angle between camera and stage should be smaller than 1°.
- The shading correction has been set up.

1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.

Selecting an objective

2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the acquisition of the stitched image.

Selecting the acquisition process

3. Activate the *Process Manager* tool window.
4. Select the *Automatic Processes* option.
5. Click the *XY-Positions / MIA* button.
   - The button will appear clicked. You can recognize this status by the button's colored background.
   - The *XY* group will be automatically displayed in the tool window.

Using the software autofocus

6. If your microscope is equipped with a motorized Z-drive, you can switch on a software autofocus.
   In the *Process Manager* tool window, click the *Autofocus* button.
   - The *Autofocus* group will be automatically displayed in the tool window.
7. In the *Autofocus* group, select the *Multiposition / MIA* check box.
8. If the sample surface is not plane or if it is inclined to the objective, choose the *Every MIA frame* option. Now, the software autofocus will be performed before every image acquisition.

Putting the stage navigator on display

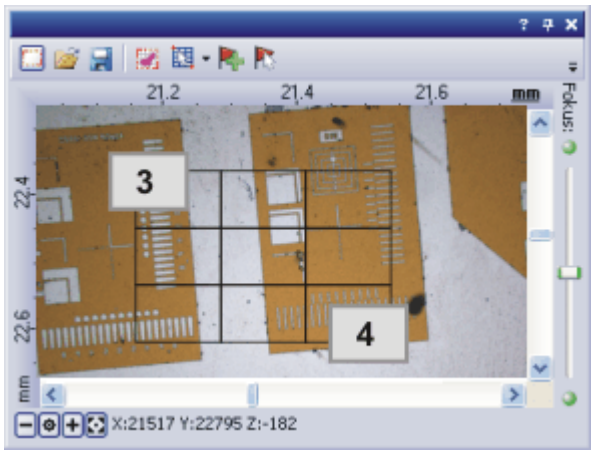
9. In the *Process Manager* tool window, click this button .
   - The *Stage Navigator* tool window is shown. When you have acquired an overview image of your sample, you will see this area of the image in the stage navigator's image segment.
10. Set the magnification for the image segment in the *Stage Navigator* tool window. To do this, use the zoom buttons at the bottom left of the tool window (**2**).
    The current stage position will be shown by a yellow rectangle in the image segment (**1**). You should choose a magnification that enables you to see this

rectangle clearly.

- You can find more information on the *Stage Navigator* tool window in the online help.

Defining the MIA scan area

11. In the *Process Manager* tool window, click this button .
    - The system will automatically switch into the live mode.
    - The *Define MIA Scanning Area* dialog box opens.
12. Move the XY-stage to the top left-hand corner of the MIA scan area you want (**3**).
13. Focus, then select the optimal settings for your acquisition, in the *Camera Control* tool window. Pay special attention to setting the correct exposure time. This exposure time will be used for all of the stitched image's individual images.
14. Confirm the starting position in the *Define MIA Scanning Area* dialog box with *OK*.
15. Move the XY-stage to the bottom right-hand corner of the MIA scan area (**4**). Confirm this position in the *Define MIA Scanning Area* dialog box with *OK*.
    - In the *Stage Navigator* tool window, the MIA scan areas that have been defined are displayed. Here, you can immediately see how many individual images are required for the acquisition of the stitched image, when the current magnification is used.



Acquiring a stitched image

16. Click the *Start* button.

- The acquisition begins immediately. The individual images are acquired, then immediately assembled. You can watch how the stitched image grows, in the image window.
- In the status bar at the bottom left of the user interface, you will find a progress bar, the number of images already acquired, and the total number of frames (e.g., 3/9).
- The acquisition has been completed when you can once more see the *Start* button in the *Process Manager* tool window, and the progress bar has been faded out.
- You see the completed stitched image, in the image window. The individual images won't be saved separately.
- The stitched image will, by default, be automatically saved. The storage directory is shown in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.

Acquiring a stitched image with extended depth of focus

The acquisition of a stitched image with extended depth of focus, is both with and without, a motorized XY-stage, possible.

Without a motorized XY-stage

1. Start the *Manual MIA* acquisition process.

   You can find a step-by-step instruction for doing this further above.

2. Click the *Instant EFI* button, in the *Manual MIA* group.
   - The *Instant EFI* acquisition process will start at once. Instead of the live-image, you now see the EFI image.
3. Now move your microscope's Z-drive slowly and change the focusing of the image. Observe how the EFI image builds itself up.
   - For each image that is acquired, the sharpest image segment is adopted in the EFI image.
4. When all of the image structures are sharply displayed, click one of the direction arrows in the *Manual MIA* group to continue with the acquisition of the stitched image.

Note: You now see the live-image with the last focus settings. That means that normally, the live-image won't be in focus.

5. Bring the image into focus.
6. Repeat the last steps for each of the stitched image's individual images for which you want to use the *Instant EFI* acquisition process.
7. Click the *Stop* button when you want to end the acquisition of the stitched image.
   - You see the completed stitched image, in the image window.

With a motorized XY-stage

You can only use the *EFI* acquisition process when your stage is equipped with a motorized Z-drive.

1. Select the *XY-Positions / MIA* acquisition process.
2. Define an MIA scan area.
   You can find a step-by-step instruction for doing this further above.

3. Additionally, select the *Z-Stack* acquisition process.
   - In the group with the different acquisition processes, two of them are now active:

4. Define all of the parameters for the Z-stack's acquisition.
5. In the [*Z*] group, select the *Extended Focal Imaging* check box.
6. Click the *Start* button to begin the acquisition of the stitched image.
   - At each of the MIA scan area's stage positions, a Z-stack will first be acquired, then the EFI image calculated from it. The EFI images will be combined into a stitched image.

- When the acquisition process has been completed, you'll see the finished stitched image in the image window.

Automatically acquiring several stitched images

You can define several MIA scan areas on the sample. When the acquisition has started, all of the MIA scan areas will be moved to, one after the other, and a stitched image will be acquired at every position.

1. Select the *XY-Positions / MIA* acquisition process.
2. Define several MIA scan areas. You can find a step-by-step instruction on how to define an MIA scan area further above.
   Begin with the area of the sample that is to be scanned first.

Putting the stage navigator on display

3. In the *Process Manager* tool window, click this button .
   - The *Stage Navigator* tool window is shown. When you have acquired an overview image of your sample, you will see this area of the image in the stage navigator's image segment.
   - In the *Stage Navigator* tool window, the MIA scan areas that have been defined are displayed. They are numbered serially in the order in which they were defined.

Acquiring stitched images

4. Click the *Start* button to begin the acquisition of the stitched image.
   - Each of the MIA scan areas will now be scanned, and the stitched image created. The scan areas will be scanned in the order that is predefined by the numbering.
   - All of the stitched images will be acquired with the current camera, and current acquisition settings.
   - When the acquisition process has been completed, you'll find a stitched image for each of the MIA scan areas, in the document group.

Combining individual images into a stitched image

Use the *Process > Multiple Image Alignment…* menu command to have several separate images combined, as with a puzzle, into a stitched image. The individual images will be combined in their full X/Y-resolution. The stitched image will thus display a large sample segment in a higher X/Y-resolution than would be possible with a single acquisition.

Acquiring images

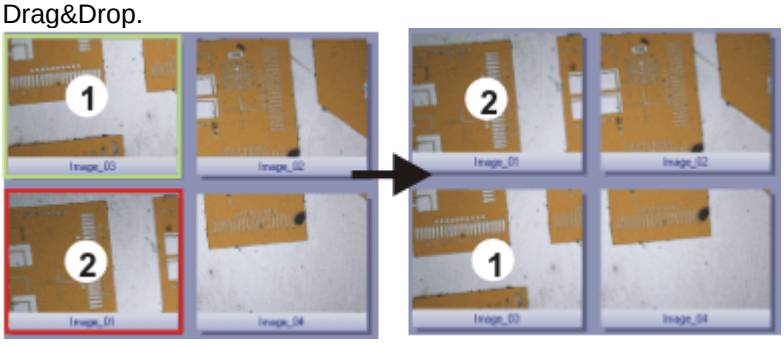
1. Load the images you want to combine, or acquire a suitable set of images.
   - All of the images you want to combine must be of the same image type. You can't, e.g., have a gray-value image combined with a true-color image.
   - When you acquire the images, number their names sequentially, e.g., "Image001", "Image002" and so on. In many cases, the images will then already be arranged in the right order in the *Multiple Image Alignment* dialog box.

Selecting images

2. Open the *Gallery* tool window. To do this, use, e.g., the *View > Tool Windows > Gallery* command.
3. Select all of the images you want to combine in the *Gallery* tool window.

Assembling images

4. Use the *Process > Multiple Image Alignment...* command. This command is only active when more than one image of the same image type has been selected.
   - The *Multiple Image Alignment* dialog box opens.
   - The dialog box's stitching area will display a preview of the individual images.
5. If necessary, while keeping your left mouse button depressed, drag on the bottom left-hand corner of the dialog window to enlarge it. Alternatively, double click the header of the dialog box to enlarge the dialog box to full-screen size.
6. Check whether the images' positions are correct. You can change the arrangement of the individual images, e.g., by exchanging two images in the stitching area by Drag&Drop.



   - The illustration shows the stitching area with four individual images. On the left, the images 1 and 2 are not in the correct position. Image 1 (green frame) will therefore be dragged onto image 2 (red frame). On the right, you see the stitching area after the two images have been interchanged.
7. When the individual images overlap, select the *Correlation* option in the *Output > Alignment* list.
   Then your software will search for the same image structures in neighboring

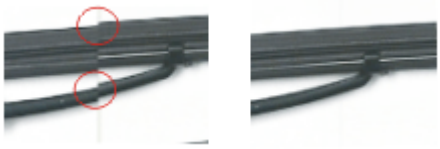
individual images. The stitched image will be put together in such a way that image areas that are the same will be superimposed.

8. Click the *OK* button to carry out the automatic image alignment.
   - The *Multiple Image Alignment - Manual Align* dialog box opens.
   - The stitched image will be displayed.

Checking a stitched image

9. Check the stitched image on display.
   Use the zoom buttons in the dialog box to zoom in the stitched image in the dialog box.

10. Should individual images have been incorrectly assembled, you can manually shift one or more of them, in respect to one another.
    To do this, click in the image you want to shift, then drag it with your left mouse button depressed, in the required direction.
    - The currently selected image will be displayed semi-transparently to make it easier for you to find the point of contact with the neighboring image.



    - Two images were not correctly aligned with each other. There is a misalignment. When the manual alignment has been made, the two images fit together seamlessly.
11. Select the *Cut Edges* check box to clip the image in such a way that there are no longer any empty areas visible on its borders.
    - In the preview, the image edges that are to be clipped will be displayed semi-transparently.
12. Select the *Equalize* check box if the images aren't homogeneously illuminated. Then the intensity values of the individual images will be matched with one another, which will make the background appear more homogeneous.
13. Click *OK*.
    - A new image with the name "Image_<consecutive No.>" will be created.

00383

5. Processing images

5.1. Commenting images

There are several different ways of adding notes to an image.

Using drawing objects

The *Drawing* toolbar makes a variety of drawing functions (line, rectangle, ellipse, text) available to you, as well as options for color selection and line styles.

You can find more information on working with drawing objects in the online help.

Using annotations

You can use the *Annotations* tool window to mark interesting positions in an image, to name them and to save them. You can give each position a text or audio annotation. In this way you will be able to jump to the position in the image that you want with one mouse click, and this will be immediately shown in the magnification you want.

Use this possibility especially when you are commenting on very large images.

You can find more information on working with positions and views in the online help.

Entering an image comment

The *Properties* tool window will show you all of the available information from the document group on the active image.

You can also supplement this information with a text annotation of your own on the image. Enter your comment in the *Note* field. You can find step-by-step instructions in the online help.

00121

5.2. Processing images

The *Process* menu offers numerous image processing functions, with which you can change an acquired image (e.g., increase the image contrast or the image sharpness).

1. Load the image you want to process, or activate the image in the document group.
   - Please note that the *Process* menu will only be visible when an image window is active in the document group.
2. Use one of the commands in the *Process* menu, e.g., *Process > Enhancement > Adjust Intensity...* .
   - The image processing dialog box will open. The image processing operation that is active will be shown in the dialog boxes header.
3. Click the small arrow next to the *Preview* button to open a list of all of the preview functions. Select the *Original and Preview* entry.
   - This preview function displays the same image segment twice in the dialog box. The first one shown is the source image. The second is the image that results when the current parameters are used.
   - Most of the image processing operations need one or two of the parameters that are shown in the *Settings* group.
4. Change the image processing operation's parameters. After every change that is made in a parameter, the operation will be immediately applied to the source image, and the resulting image will be shown in the preview window. Click the *Default* button to readopt the preset parameters in the *Settings* group, when the current parameter doesn't make sense to you.

5. When you have found the optimal parameters, click the *OK* button to have the active image processing operation applied to the image with the active parameters.
   - The image processing dialog box will closed.
   - Please note that the image processing operation changes the source image. No new image document will be created. You can, however use the *Edit > Undo* command to restore the source image.

00175

6. Measuring images

6.1. Overview

Your software offers a wide range of measurement functions. They enable you to quickly count objects and measure segments and areas. All the results will be saved together with the image and can also be issued as a sheet.

Prerequisite: For making measurements, correctly calibrated images are an essential prerequisite. Images that you have acquired with your software will have been automatically correctly calibrated when you have specified the objective you used.
Should the image not yet have been calibrated, use the *Image > Calibrate Image...* command to carry out a calibration.

Selecting the measurement environment

Measuring with help of the tool window

Switch to the *Processing* layout when you want to measure images. You can find the *Measurement and ROI* tool window in the bottom section of this layout. In this tool window you have fast access to all measurement functions and settings which relate to the measurement. This tool window is at the same time the measurement display and contains all of the values that have been measured on the active image.

Note: Should, right at the bottom of the user interface, several tool windows lie one over the other, activate the *Measurement and ROI* tool window, by clicking on the header of the *Measurement and ROI* tab. The tabs can be found under the tool windows.

Starting a measurement

Begin a measurement by selecting the measurement function you want. You will find the measurement function in the *Measurement and ROI* tool window, on the *Measurement and ROI* toolbar, or in the *Measure* menu.

Working in the measurement mode

As soon as you have clicked a measurement function, your software will automatically switch to a measurement mode. In the measurement mode your mouse pointer will take on the shape of a cross on the image. You can make as many measurements as you like with the measurement function that has been selected. The continuous measurement mode is valid for all loaded images. You can, therefore, easily measure numerous images one after the other.
The selected measurement function's button will keep its clicked appearance and in this way show you the current measurement function. You can recognize this status by the button's background color.

Finishing the measurement mode

You will remain in this measurement mode until you explicitly switch it off. To do so, click the *Select Measurement Objects* button. You can find the button either in the *Measurement and ROI* tool window or on the toolbar.

Changing the default measurement mode

The continuous measurement mode described above is preset by default. You can change this default setting. To do this, use the *Tools > Options...* command. Select the *Measurement and ROI > General* entry in the tree view. Select the *Switch to 'Select Measurement Objects' mode after creating a measurement object* check box. Then, when you have completed a measurement, you will automatically leave the measurement mode again. This means you have to select the measurement function again before you start each interactive measurement.

Displaying and saving measurement results

The measurement results will be displayed directly on the image and in the *Measurement and ROI* tool window. Use the *View > Tool Windows > Measurement and ROI* command to have the tool window displayed.

Saving the measurement results

The measurements will be saved along with the image, if you save the image in the TIF or VSI file format.

You can, however, also export the measurement results in a results sheet, and save this as a file. To do this, use the *Export to Excel* or *Export to Workbook* command. You find both of the commands, e.g., in the *Measure* menu.

Showing and hiding measurement results in an image

The measurement results will be shown on the image in a special data layer, the measurement layer. On your monitor, image and measurement layer are shown together. The data of each, however, is individually stored if you use the TIF or VSI image file format. Try and picture the measurement layer as a transparency which is placed over the image. When you measure an image, the image data will not be changed by having the measurement results displayed on it.

You can, at any time, hide or show the measurement layers.
To do so, use the *Layers* tool window. There you have access to all of an image's layers. The eye icon identifies all of the layers that are currently on display on your monitor.
Click the eye icon in front of the measurement layer to hide the measurements. Click an empty cell without an eye icon to make the corresponding layer reappear.

Editing measurements

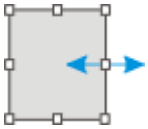
You can edit existing measurement objects at any time. To do so, click the *Select Measurement Objects* button. You can find the button either in the *Measurement and ROI* tool window or on the toolbar.

Note: When you load an image file with measurement objects, it is only possible to edit the measurement objects if the image file has been saved in the TIF or VSI image file format.

Moving measurement objects

You can move a measurement object while keeping the left mouse button depressed.

Increasing/decreasing the size of measurement objects



You can also change the size of a measurement object. Move the pointer onto a marker. By dragging the marker with the mouse button depressed, you can adjust the frame's size as wished.

Deleting measurement object

Click the [Del] key on your keyboard in order to delete the selected measurement object.

What happens to the measurement values?

The measurement values in the *Measurement and ROI* tool window will be correspondingly updated.

Changing the color, font and line thickness of individual measurement objects

You can, at any time, change the color, font, and line thickness, of individual measurement objects. Select one or more measurement objects in an image and click your right mouse button to open a context menu. In the context menu you'll find several commands with which you can change the appearance of the selected measurement objects.
From the context menu, select, e.g., the *Change Color...* command. Select the color you would like from the color palette, then close the dialog box with *OK*.

Measuring in the live mode

All of the measurement functions are also available in the live-image. You can therefore, e.g., quickly measure a segment in the live-image.

Measuring on different image types

Measuring on multi-dimensional images

You can combine a series of individual images into one image. What results is e.g., a time stack in which all of the frames will have been acquired at different times.

You can make measurements on every separate image. Display the required frame on your monitor. To do this, use the navigation bar in the image window. Then carry out the measurement on this frame. The measurement will be permanently linked to this frame, i.e., the measurement will only be displayed on your monitor when the frame on which you made this measurement is also on display.
The measurement results will be shown in the *Measurement and ROI* tool window. You can give every measurement the number of the frame on which it was made. To do so, use, e.g., the measurement parameter *Index t* for time stacks.

Measuring on multi-layer images

With some functions, e.g., with the *Image > Combine Color Images...* function, a multi-layer image will be created. This multi-layer image is made up of several layers. You can find more information on multi-layer images in the online help.

Measurements always apply to one image layer. For this purpose, show the image layer on your monitor, on which you want to make measurements. To do so, use the *Layers* tool window. Then carry out the measurement on this image layer. The measurement will be permanently linked to this image layer, i.e., the measurement will only be displayed on your monitor when the image layer on which you made this measurement is also on display.
The measurement results will be shown in the *Measurement and ROI* tool window. You can give every measurement the name of the image layer on which it was made. To do this, use the *Layer* measurement parameter.

Measurement precision

How precise the measurement is, depends on the X/Y-calibration and the image's current zoom factor.

Influence of the X/Y-calibration

The X/Y-calibration defines the width and height of the sample area that is represented by one pixel. For example, it could be that one pixel displays a sample area of 10 µm x 10 µm. A pixel is the smallest image structure that can be measured. For this reason, the maximum measurement precision where this example is concerned, is 10 µm.

Influence of the zoom factor

The zoom factor tells you how large the image will be displayed on your monitor. With a zoom factor of 100%, one pixel on the monitor equals exactly one pixel in the image. With a zoom factor of 50%, one pixel on the monitor equals 2 x 2 pixels in the image. When you make a measurement, you should use the zoom factor 100% whenever possible. Then you will achieve a maximum of measurement precision. Should the zoom factor 100% not be possible, because the image area you want to measure can't then be completely seen, choose the largest possible zoom factor under 100%.
Information about changing an image's zoom factor can be found in the online help.

00150 02072012

6.2. Measuring images

Your software offers a wide range of measurement functions. They enable you to quickly count objects and measure segments and areas.

The following step-by-step instructions present the measurement functions to you by way of several examples.

Measuring image objects interactively

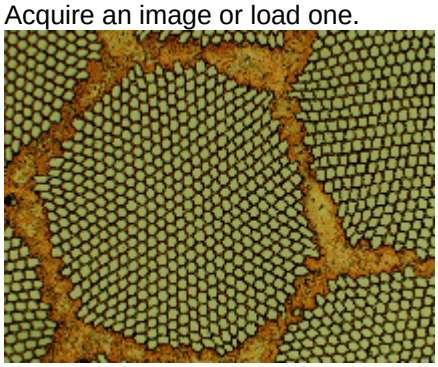
Task

You want to measure the filaments in a supraconductor.
To do this, load a suitable image, or acquire one. Measure the diameter of several of the hexagonal filaments, in each case between the opposing vertices. Subsequently edit the measurement. Delete some of the measurements you've made. Enter the results in a MS-Excel sheet.

1. If necessary, use the *View > Tool Windows > Measurement and ROI* command to have the *Measurement and ROI* tool window displayed.
   - You'll find the tool window at the lower edge of the user interface. It's possible that it may be covered by the *Count and Measure Results* tool window. Click the *Measurement and ROI* tab at the bottom of the user interface to bring the tool window into the foreground.

Loading an image

2. Acquire an image or load one.



During the installation of your software some sample images have been installed, too. You can follow these step-by-step instructions when you use the example image "SupraConductor.tif".

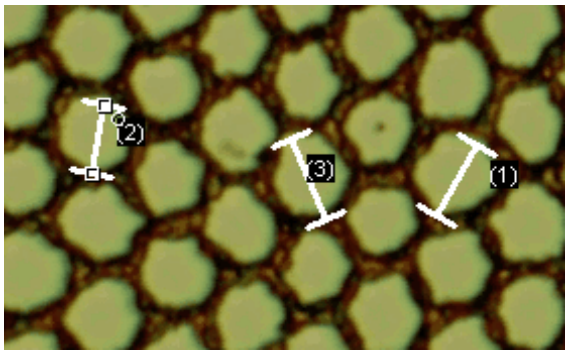
Setting the labeling color

The measurement results will be written into the image according to the default settings, in red font color and without a background. This can't be easily read against the superconductor's structure. Change the labeling settings.

1. Use the *Tools > Options...* command.
2. Click the *Measurement and ROI > Measurement Display* entry in the tree view.
3. Click in the *Background Color* field, and choose, e.g., the color Black.
4. Select the *Text color > Fixed colors* option, then select the color White from the palette, to see the measurements in white and the labeling in black, in the image.
5. Close the dialog box with *OK*.

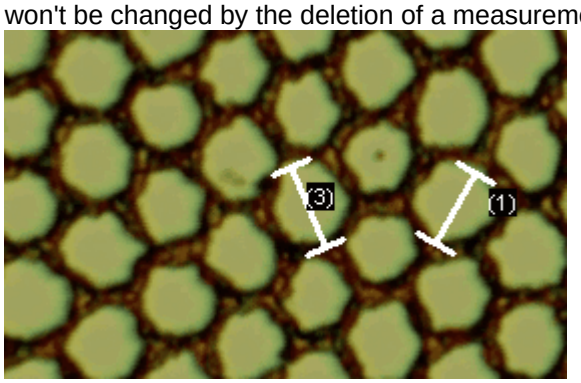
Measuring lengths

1. Click the *Arbitrary Line* button, located on the toolbar at the top of the tool window.
2. Click with your left mouse button at the starting point and end point of the reference distance.
3. If you have measured a reference distance, you can immediately proceed with the next measurement.
4. Click the *Arbitrary Line* button again, to end the length measurement.
5. Take a look at the results in the tool window and in the image.
   - The illustration shows the image with three executed measurements. The measurement 2 has been selected.



Deleting measurements

1. Click one of the measurement results in the *Measurement and ROI* tool window.
   - The corresponding line will be marked in the image.
2. Press the [Del] key.
   - The measurement will be deleted both in the image and in the tool window.
   - When a measurement has been deleted, the image and the tool window contain one measurement less. The IDs of the remaining measurements won't be changed by the deletion of a measurement.



When you've completed the measurements, you should switch off the measurement mode, since otherwise, you might inadvertently select your measurements and move them.

3. Check whether one of the buttons on the *Measurement and ROI* tool window's toolbar appears clicked. Release this button

Exporting results to MS-Excel

1. To do this, click the *Export to Excel* button.
2. In the In/Output dialog box you set up the directory in which the data is to be saved, and enter the name of the MS-Excel sheet. Adopt the file type "Excel-Sheet (*.xls)“.
3. Click the *Save* button to have the MS-Excel sheet with the measurement results saved.

Closing the image

1. Click the *Close* button, located at the top right of the document group.
   - You have changed the image because you've added interactive measurements. For this reason, you'll receive a query whether you wish to save the image or not.
2. Save the image in the TIF or VSI file format. The measurements will then also be saved in the image file. They can at any time, be edited deleted or augmented.

Outputting various measurement parameters

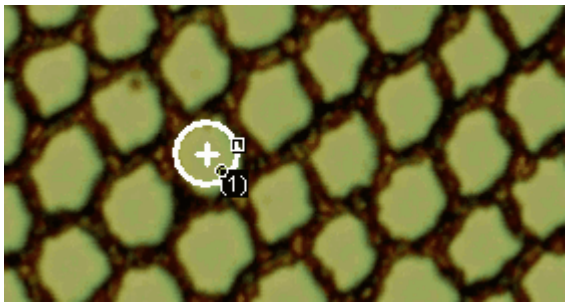
Task

You want to measure the filaments in a supraconductor.
Measure the hexagonal structure as a circular surface. Have a variety of measurement parameters, such as the area, the perimeter and the diameter, output. Have the diameter shown in the image.

1. Acquire an image or load the "Supraconductor.tif" example image.

Measuring areas

2. In the *Measurement and ROI* tool window, click the *2 Points Circle* button.
3. With your left mouse button, click in the center point of the hexagonal structure.
4. Move your mouse, and in the process drag out the circle. Match the circular object as well as possible to the structure. Click the left mouse button.
5. Click the *2 Point Circle* button again, and switch off the measurement mode.
6. Take a look at the result in the *Measurement and ROI* tool window.
   - The illustration shows the image of the superconductor with a circular measurement.



Viewing the list of measurement parameters

1. In the *Measurement and ROI* tool window, click the *Select Measurements* button.
   - In the dialog box you'll see a list with all of the available measurement parameters. At the bottom of the dialog box you'll see a list of the measurement parameters that are calculated for all objects.

- A detailed description of this dialog box can be found in the online help.

Outputting additional measurement parameters

2. Go to the list of all of the available parameters, then click the "Diameter" measurement parameter.
   - On the right, an illustration shows you how the parameter is calculated.

   You can see that there are different ways in which the diameter of a 2D object can be calculated.
3. Click the "Mean" entry in the list under the illustration to select the "Mean (Diameter)" measurement parameter. When you do this, the mean value of all of the possible diameters is determined.
4. Click the *Add 'Mean (Diameter)'* button.
   - This measurement parameter will be adopted in the list. All of these measurement parameters will be displayed in the tool window.
5. Close the dialog box with *OK*.
6. Take a look at the result for the circle's diameter in the *Measurement and ROI* tool window.

Outputting measurement parameters in the image

1. Open the *Select Measurements* dialog box.
2. At the bottom of the list of all of the calculated measurement parameters, click the "Mean (Diameter)" measurement parameter.
3. To the right of this list you'll see a button with a blue arrow . Click this button to move the measurement parameter to the top of the list.
4. Close the dialog box with *OK*.
5. Take a look at the result for the circle's diameter in the image.

Note: The measurement display in the image has to be updated once, so that the settings that have been changed are also taken into account. You update the measurement display, for instance, by adding another measurement, or by once selecting an existing measurement in the image.

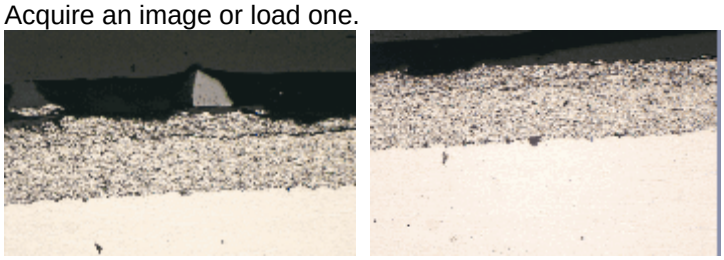
Measuring several images

Task

You want to measure the thickness of a spray coating. To do so, you acquire several images of the coating. Have the results from all images displayed simultaneously. Take a look at the mean value for all of the measurements.

Loading an image

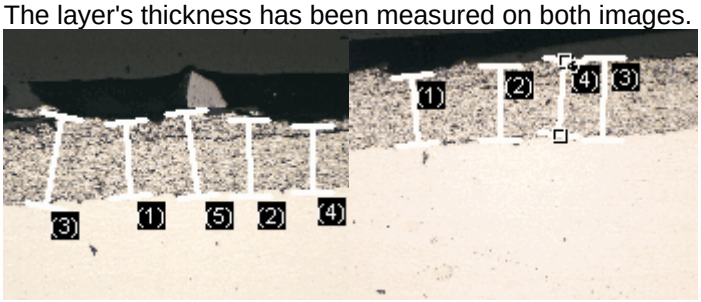
1. Acquire an image or load one.



During the installation of your software some sample images have been installed, too. You can carry these step-by-step instructions out directly with the example images "SprayCoating2.tif" and "SprayCoating4.tif".

Measuring the layer thickness

2. Activate the first image in the document group.
3. Click the *Arbitrary Line* button, located on the toolbar at the top of the *Measurement and ROI* tool window. Measure the thickness of the layer at several different places.
4. Activate the next image. Measure the thickness of the layer at several different places, here also.
5. Click the *Arbitrary Line* button again, and switch off the length measurement.
   - The layer's thickness has been measured on both images.



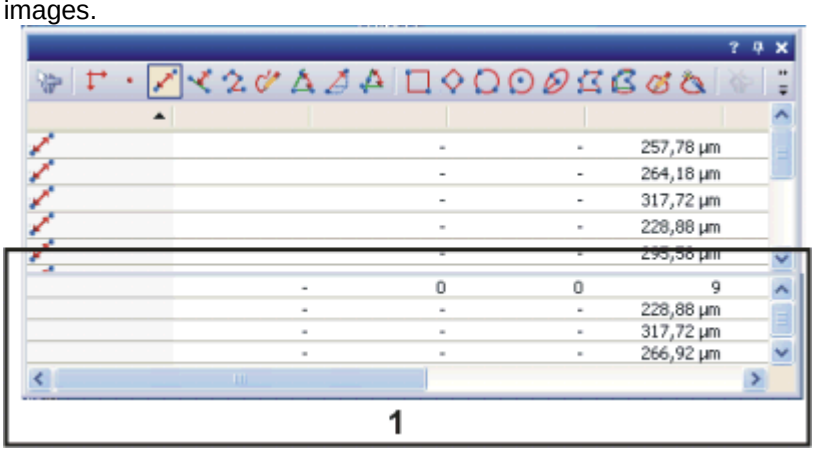
Displaying the measurement results of all of the images

1. In the *Measurement and ROI* tool window, click the *Measurement and ROI Options* button.
2. Select the *Measurement and ROI > Results* entry in the tree view.
3. Clear the Show measurement objects: Only of the active image check box.
4. Close the dialog box with *OK*.
   - Now the results for both images will be shown simultaneously in the tool window.

Viewing the statistical parameter

1. In the *Measurement and ROI* tool window, click the *Measurement and ROI Options* button.
2. Select the *Measurement and ROI > Results* entry in the tree view.
   - In the *Statistic* group, you'll find various statistical parameters.
3. Select the *Mean* check box.
4. Close the dialog box with *OK*.

- Now, in the *Measurement and ROI* tool window under the measurement results, the chosen statistical parameter (1) will by shown. You can see there the mean value of the layer thickness for all of the measured images.

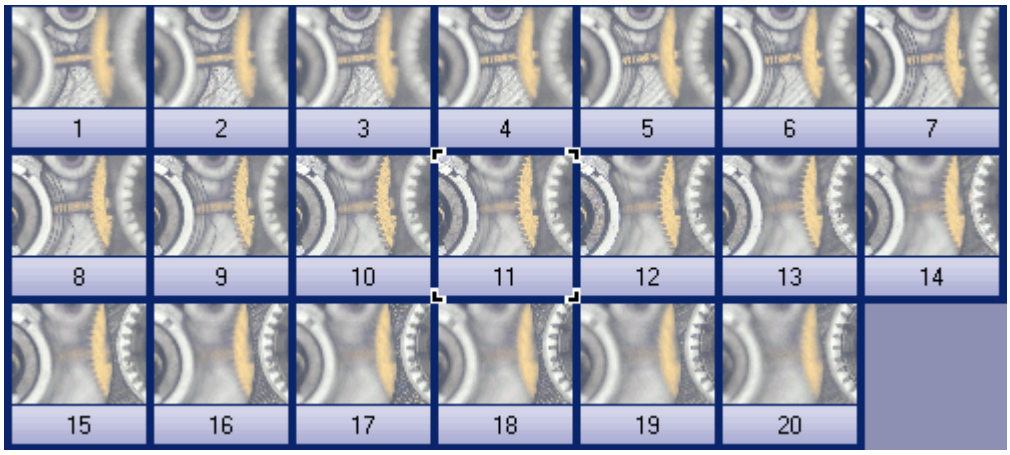


Measuring heights

Task

To be able to make height measurements, you need to have a height map. A height map is a gray-value image whose gray values contain height information. The multi-dimensional image "Clockwork" is a Z-stack. Use the EFI algorithm to calculate the height map. Measure the height difference between the brass-colored gear wheel in the middle image segment, and the silver-colored gear wheel on the right-hand side of the image.

1. Load the "Clockwork.tif" image.
   - The "Clockwork.tif" image is a Z-stack. The works of a clock was analyzed under a reflected light microscope. In the process, images of the works were acquired at different focus positions.
     In the illustration, the Z-stack is displayed in the tile view. Pay attention to the gear wheel that is only sharply reproduced in the middle of the Z-stack.



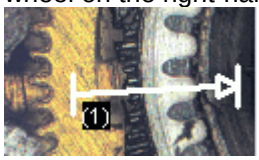
Creating a height map

2. Use the *Process > Enhancement > EFI Processing…* command.
3. Select the *Apply on > All frames and channels* option.
4. In the *Algorithm* list, select the *Reflected light* entry.
5. Select the *Height map* check box.
6. Select the *Create new document as output* check box.
7. Close the dialog box with *OK*.

- You can see the EFI image with the clockwork's texture. The resulting image is a multi-layer image and is therefore accompanied by this icon in the image window's title.
- The height map is a layer of the EFI image. The texture image makes up the second layer. The height information is therefore also present in the EFI image. You can measure the height directly on the texture image.

Measuring height

1. In the *Measurement and ROI* tool window, click the *3D Line* button.
2. Now measure the height between two image objects.
   Click, for example, the brass-colored gear wheel, and the silver-colored gear wheel on the right-hand side of the image.



3. Click the *3D Line* button in the *Measurement and ROI* tool window again, then switch off the 3D measurement.
   - In the *Measurement and ROI* tool window, and in the image, the "3D Length" measurement parameter is output with the line's complete length.
     The "3D Intensity Projection" measurement parameter measures the height difference between two points.

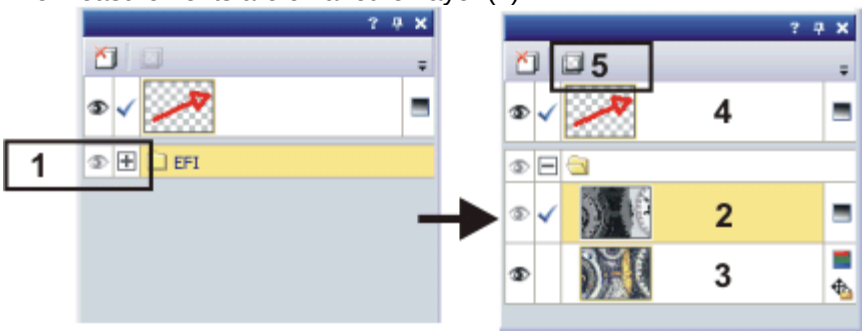
Issuing the height difference

1. In the *Measurement and ROI* tool window, click the *Select Measurements* button.
2. In the list of all of the available measurement parameters, take a look at the parameter of the "3D Line" object type.
3. Select the measurement parameter „3D Intensity Projection“.

4. Insert this measurement parameter into the list of the displayed measurements.
5. Close the dialog box with *OK*.
   - In the tool window, you'll now see the "3D Intensity Projection" measurement parameter. It tells you how far in height the two gear wheels are from each other.

Viewing the height map

1. Use the *View > Tool Windows > Layers* command to make the *Layers* tool window appear.
2. In the *Layers* tool window, click the [+] sign (**1**) and open the image's layers.
   - You can now see the image's individual layers. Height map (**2**) and texture image (**3**). The height map can't be seen, because it's absolutely transparent at the moment.
   - The measurements are on another layer (**4**).



3. Select the height map in the *Layers* tool window.
4. Click the *Set Layer Opacity* (**5**) button, located on the toolbar at the top of the tool window.
5. Drag the slider all the way to the right, to an opacity of "100%", then take a look at the height map.
   - Low-lying structures can be recognized by their dark gray values, structures that lie higher, by their bright gray values.

Showing/hiding layers

1. Click its eye icon to make the corresponding layer disappear. By doing this, you can, e.g., for a time, remove the measurements from the image.
2. Click an empty cell without an eye icon to make the corresponding layer reappear.

00154

6.3. Measuring welding seams

What is a weld measurement?

Measuring the cross section of a weld is a method commonly used for judging a weld's quality. With the *Weld Measurement* solution you can interactively measure microscope images of weld cross-sections and can output the results on the image and in a table. The following measurement functions are available:

	Multiple perpendicular lines	Use this measurement function, to determine the distance from several measurement points to a reference line.
	Asymmetry Lines	Use this measurement function, to construct the perpendicular bisector of the connection between two reference points, and to determine the distance of a measurement point from the perpendicular bisector.
	Throat thickness	Use this measurement function, to determine the thickness of a fillet weld's throat.

Starting a measurement

You will find the weld measurement functions in the *Measure* menu, or as a button on the *Measurement and ROI* tool window or toolbar. Start a measurement, e.g., by clicking the corresponding button.

Interactive measurement functions and weld measurements

The functions which you can use to measure welds, behave just like the other interactive measurement functions offered by your software, e.g. the *Arbitrary Line* measurement function. All of the information on the interactive measurement functions also applies for the measurement of welds.

10802

Measuring a throat thickness

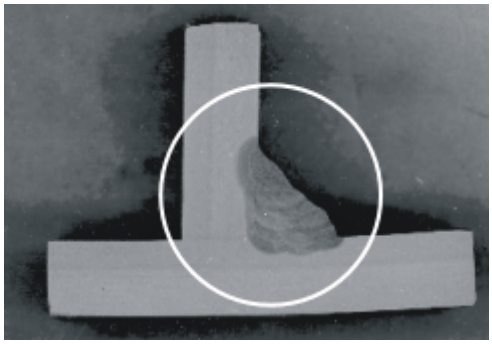
Use the *Throat thickness* measurement function, to determine the thickness of a fillet weld's throat. You can find the measurement function in the *Measure* menu, or as a button on the *Measurement and ROI* tool window or toolbar.

Prerequisite: The *Throat Thickness* measurement function will only be available if you purchased the *Weld Measurement* solution together with your software.

1. If necessary, use the *View > Tool Windows > Measurement and ROI* command to have the *Measurement and ROI* tool window displayed.

Loading an image

2. Acquire an image or load one.

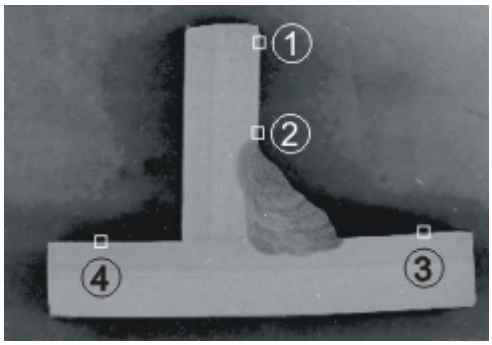


The illustration shows a cross section of two welded pieces of metal. The weld is circled.
How thick is the weld's throat?

3. Set a zoom factor for your image window that will make the image segment that is to be measured clearly visible. You will achieve the most precise measurements if you set the zoom factor at 100%.

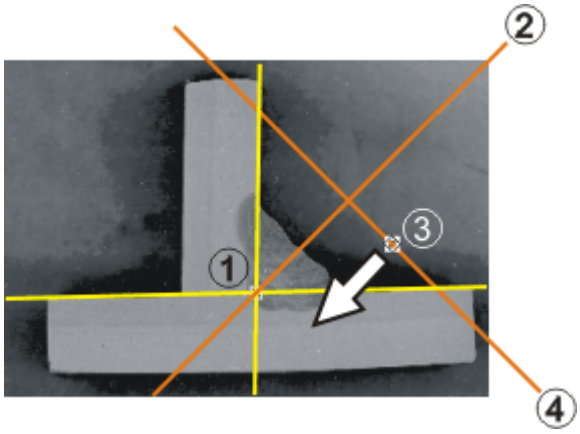
Measuring the throat thickness (with a welded root)

4. Start the measurement. To do so, click the *Throat Thickness* button, on the toolbar at the top of the tool window.



With four mouse clicks (**1**-**4**), define two lines along the inner surfaces of the metal pieces which are welded together.

5. Click on a point on the inner surface of the first piece of metal (**1**). This point should be a far as possible from the weld's root. You can put the measurement point before or after the weld.
   - The point which you define will be shown in the image with a handle.
   - The mouse pointer's shape on the image window shows which measurement mode you are in.
6. With three more mouse clicks (**2**-**4**), define two lines along the inner surfaces of the metal pieces which are welded together.
   - Your software will now automatically display some lines and handles in the image window.
   - The mouse pointer is now linked to an auxiliary line, which is perpendicular to the bisector of the angle. When you move the mouse you will move this line at the same time.

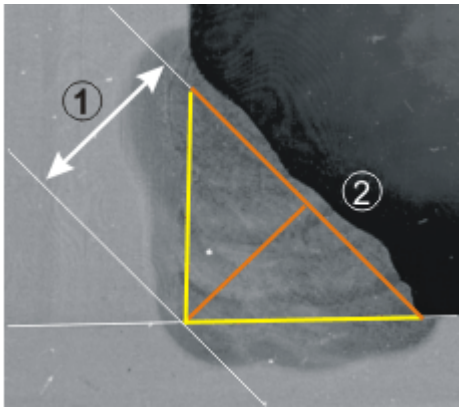


Once the inner surfaces are defined (yellow lines), the position of the root (**1**) will automatically be drawn in and your software will calculate the bisector of the angle (**2**). Together with the handle (**3**), move the line (**4**) which is perpendicular to the bisector of the angle to determine the throat thickness.

7. Move the auxiliary line (**4**) to the outer surface of the weld. The part between the two lines shown in yellow above must be just inside the cross-section of the weld along the entire length.
   - The measurement of the throat thickness is thus complete. The *Throat Thickness* measurement object (an equilateral triangle) is completely defined.
   - The throat thickness (the height of the triangle) will be shown in the image. In the *Measurement and ROI* tool window table, a new measurement value with a type of *Throat Thickness* will be entered.

Note: If the measurement results are not shown, check the measurement parameters currently displayed. You can find step-by-step instructions to modify the measurement parameters further down.

8. Take a look at the result in the *Measurement and ROI* tool window and in the image.



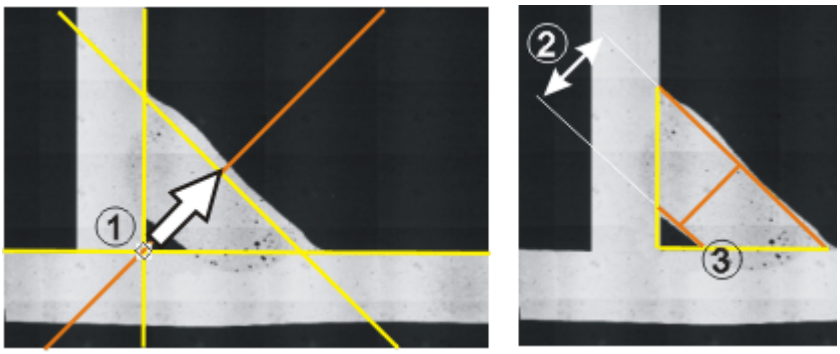
The illustration shows an enlargement of the weld with the measurement object (an equilateral triangle). The result of the measurement is the throat thickness (**1**) (the height of the triangle). The auxiliary line (**2**) (the base of the triangle) must be positioned so that it just lies completely within the weld.

Measuring the throat thickness (when the root is exposed)

If the root of the weld is exposed, another step will be required to measure the throat thickness.

9. Select the measurement object.

   To do so, click the *Select Measurement Objects* button to switch to a selection mode and then click on the measurement object in the image window. You can find this button, e.g., on the *Measurement and ROI* tool window's toolbar.
   Directly after a throat thickness measurement, the measurement object will automatically be selected.
10. Click on the vertex.
11. Keep the left mouse button pressed and drag the vertex towards the outer seam of the weld towards the base of the triangle). In this way you are moving a second auxiliary line. This auxiliary line must also lie with its full length just within the weld's cross section.



If the root of the weld is exposed, drag another auxiliary line (**3**) from the vertex (**1**). The throat thickness is now the distance (**2**) between the two auxiliary lines which are perpendicular to the bisector of the angle.

Saving the image

12. Save the image in the TIF or VSI file format. The measurements will then also be saved in the image file. They can at any time, be edited deleted or augmented.

Finishing the measurement

13. You can now measure other images.
14. If the *Throat Thickness* button is still active, click on the button again to finish measurement mode.

Changing settings for a throat thickness measurement

Adjusting measurement parameters

During each interactive measurement, significantly more values are measured than can be shown in the image or in the *Measurement and ROI* tool window. To alter the measurement parameters shown, follow these step-by-step instructions. In particular, ensure that at least the *Length* and *Angle* measurement parameters are displayed, as they are both used for the throat thickness measurement.

1. In the *Measurement and ROI* tool window, click the *Select Measurements* button.
   - A detailed description of this dialog box can be found in the online help.
2. In the *Available measurements* list, click on the *Measurement* column title to alphabetically sort all of the parameters.
3. Select the *Length* measurement parameter in the *Available measurements* list. This measurement parameter corresponds to the throat thickness.
4. Click the *Add 'Length'* button, to have the *Length* measurement parameter added to the list of calculated measurement parameters.
5. Also add the *Angle* parameter to the list of calculated measurement parameters.
6. You can now further modify the display of the measurement parameters for a throat thickness measurement. You can for instance delete all other measurement parameters currently shown, so that the list of the measurement results becomes clearer.
7. Close the dialog box with *OK*.
8. Do a throat thickness measurement and examine the result in the *Measurement and ROI* tool window.

Displaying measured angles in addition to the throat thickness in the image

By default, for a throat thickness measurement, the measurement will be shown in the image. You can also output the angle between the two pieces of metal which have been welded together in the image.

1. Carry out a throat thickness measurement or load an image which contains a throat thickness measurement.
2. Select the measurement object on the image. For example, select the corresponding measurement in the *Measurement and ROI* tool window.
3. Click the right mouse button and select the *Create Angle* command in the context menu.
   - In addition to the throat thickness, the angle measured will also be shown in the image.
   - This command creates another measurement object of type *Angle*. In the *Measurement and ROI* tool window you will thus see two entries for the measured weld.

Note: The measurements on a screen will automatically be numbered in sequence. The angle measurement will thus always have a different measurement ID from the associated throat thickness measurement. You can switch off the display of the measurement IDs, if you find their display distracting. To do so, open the *Tools > Options > Measurement and ROI > Measurement Display* dialog box, and clear the *Show ID* check box.

- With the commands in the context menu you can change the color of the labels, of the measurement lines, the reference lines or you can even alter the font.

4038

Performing an asymmetry measurement

Use the *Asymmetry Lines* measurement function to construct the perpendicular bisector of the connection between two reference points and to determine the distance of a measurement point from the perpendicular bisector. You can find the measurement function in the *Measure* menu, or as a button on the *Measurement and ROI* tool window or toolbar.

Prerequisite: The *Asymmetry Lines* measurement function will only be available if you purchased the *Weld Measurement* solution together with your software.

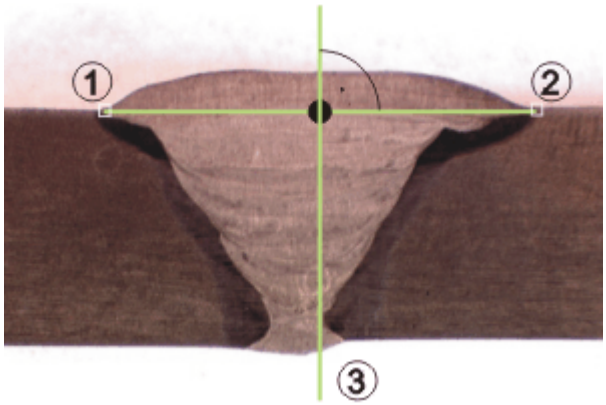
1. If necessary, use the *View > Tool Windows > Measurement and ROI* command to have the *Measurement and ROI* tool window displayed.

Loading an image

2. Acquire an image or load one.
3. Set a zoom factor for your image window that will make the image segment that is to be measured clearly visible. You will achieve the most precise measurements if you set the zoom factor at 100%.

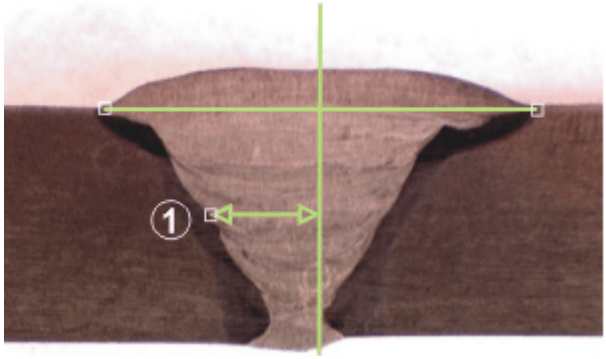
Measuring asymmetry

4. Start the measurement. To do so, click the *Asymmetry Lines* button, on the toolbar at the top of the tool window.



The illustration shows a cross section of two welded pieces of metal. How asymmetric is this weld? Click on the two reference points (**1**) and (**2**) in turn. Your software will automatically calculate the perpendicular bisector as a reference line for the measurement of the asymmetry (**3**).

5. With the left mouse button, click on two reference points in turn.
   The perpendicular bisector of the connecting line between these two reference points is the reference line for the measurement of the asymmetry. In the example shown, the reference points define the width of a weld. The reference points in the example shown lie horizontally next to one another. They could just as well have any orientation in the image.
   - The points which you define will each be marked in the image.
   - The mouse pointer's shape on the image window shows which measurement mode you are in.
   - The mouse pointer is now linked to an auxiliary line, parallel to the perpendicular bisector. When you move the mouse you will move this line at the same time.
6. Left click a measurement point to measure its distance from the reference line.
   - The result of the measurement will be shown in the image.



Define a measurement point (**1**). The distance between the point and the reference line will be measured.

7. If required, you can also define other measurement points. For each measurement point defined, the distance to the reference line will be measured.

Undoing measurement points

8. As long as the measurement is not yet finished, you can undo individual measurement points, if you have made a mistake in the measurement. To do that, press [backspace] on your keyboard.

Note: If the measurement results are not shown, check the measurement parameters currently displayed.

Finishing the measurement

9. Click the right mouse button to end the measurement.
   - In the *Measurement and ROI* tool window table, a new entry with a type of *Asymmetry Lines* will be shown.
     Please note that all of the measured distances belong to a single measurement object. In the *Measurement and ROI* tool window table there may, under certain circumstances, be several length measurements assigned to a single entry in the *Type* or *Name* column.
10. You can now measure other images.
11. If the *Asymmetry Lines* button is still active, click on the button again to finish measurement mode.

Saving the image

12. Save the image in the TIF or VSI file format. The measurements will then also be saved in the image file. They can at any time, be edited deleted or augmented.

4037

7. Performing a materials science analysis

7.1. Tool window - Materials Solutions

Use this tool window to measure an image, or several images at the same time, according to different material science analysis processes.

The *Materials Solutions* tool window works similarly to a software wizard. As soon as you've started an analysis process you'll be guided step by step through the measurement.

Overview of the supported analysis processes

(**1**) Chart Comparison
(**2**) Grains Intercept
(**3**) Grains Planimetric
(**4**) Layer Thickness
(**5**) Cast Iron
(**6**) Inclusions Worst Field
(**7**) Throwing Power
(**8**) Porosity
(**9**) Phase Analysis
(**10**) Particle Distribution
(**11**) Automatic Measurement
(**12**) Coating Thickness

Note: Which of theses analysis processes are available to you, depends on the software license you've acquired. Maybe you will only see one or two analysis processes.

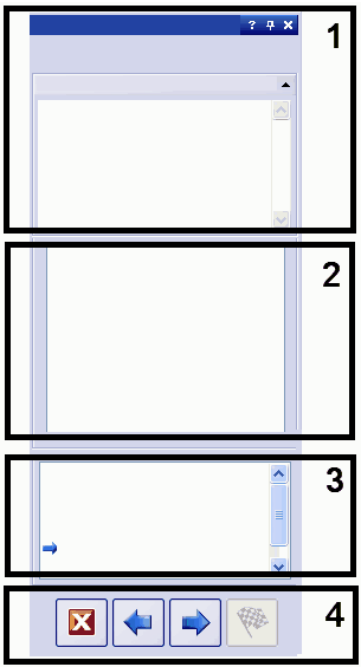
Starting an analysis process

You start an analysis process by clicking the corresponding button.

Note: A lot of your software's other functions aren't available while an analysis process is running. For example, you can't open the program options then.

Independent of which analysis process has been currently selected, the tool window is always configured in the same way. It comprises static and dynamic areas.

Makeup of the tool window



The static areas (**1**), (**3**) and (**4**) are located at the top and bottom edges of the tool window. The contents of these areas is always largely similar.

The dynamic area (**2**) is located in the middle part of the tool window. Its appearance differs according to which step and which analysis process has been chosen.

(1) Name of the analysis and "Instructions" group

You'll find the name of the current acquisition process right at the top of the tool window. In the *Instructions* group, you will find an instruction of what to do in this step and, if available, additional information.

(2) Dynamic area

The contents of this area changes completely for each analysis process and for each step in the analysis. It is therefore described each time one of the different analysis process is presented.

(3) Current step in the analysis

Here, you can see at which step in the analysis you are at this moment. The current step is indicated by a blue arrow.

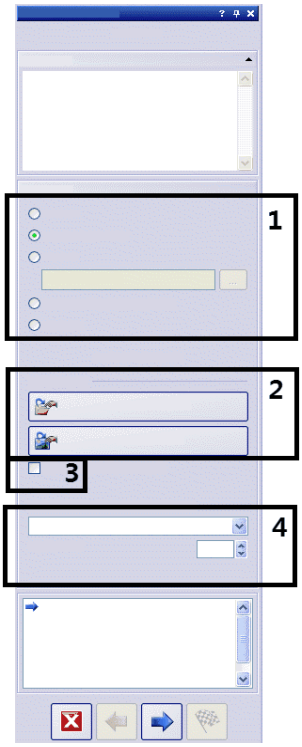
(4) Buttons

Here, you find the buttons you use to proceed to the next step in the analysis, or to return to the previous step. You can also cancel an analysis here. Depending on the current step in the analysis, not all of the buttons are active.

10242 3042013

Selecting the image source

The *Materials Solutions* tool window leads you step-by-step through a materials science measurement. In the *Image source* step, the following options are available:



(1) "Image source" group

In this group, you select the image that you want to analyze. You can also analyze several images at the same time. The following options are available:

- *Live image* option: With this option, the additional step *Image acquisition* will be shown. In this step, a live image will be acquired, which will then be analyzed in the following steps. When the *Image results* step has been completed, a new image of the live image will be automatically acquired, then analyzed. This enables you to analyze as many images as you would like during the same measurement. You can then either save the analyzed images or reject them.
- *Selected images* option: Loaded images that are currently selected in the *Gallery* tool window. Loaded images that are not selected in the *Gallery* tool window, will be ignored for the analysis.
- *Folder* option: All of the images in a specific directory. You can choose the directory as you wish.
- *Selected database images* option: All of the images you have currently selected in your software's database.
- *Stage Path* option: All images which you would like to acquire with the saved stage path. This option is only visible if your microscope stage has a motorized XY drive. You can find more information about the acquisition of images using defined stage paths in the online help.

Not all materials science analysis processes support the use of stage paths. This is why the *Stage Path* option is only available for these analysis processes: *Grains Intercept*, *Grains Planimetric*, *Inclusions Worst Field*, *Porosity*, *Phase Analysis*, *Particle Distribution*.

(2) Buttons to load saved settings

Here, you can load the settings that you want to use for the analysis. Click on the *Load from file...* button, if you want to use settings that have been saved. For example, you can in this way load the comments from a sample that has already been analyzed, and adapt them for the current sample. As well as that, with some materials science analysis processes, the slide controls that are available in the *Settings* step will also be set to the saved position.

Click on the *Get from image...* button, if you want to use the settings used for an already analyzed image for the current analysis. To make this possible, the image that has already been analyzed must be opened in your software.

(3) "Skip 'Sample Information'" check box

Select the *Skip 'Sample information'* check box to skip the *Sample information* step. As soon as you click on the *Next* button, you'll then go directly to the *Settings* step. This makes sense if you analyze numerous images of the same sample, and you only want to enter the information on the sample with the first image.

Note: When you analyze images of numerous samples, make sure the *Skip 'Sample information'* check box is not selected, because otherwise you won't see the *New Sample* button.

(4) "Check settings and results" list and "Image interval" field

This list is only of significance if you are analyzing several images. If you are only analyzing one image, leave the preset *All images* entry as it is.

If you select several images, you can choose how frequently you would like to check the settings with which the images are analyzed. If you would like to analyze a lot of images with the same settings, you can automate the analysis.

The following entries are available in the *Check settings and results* list:

- *All images*: Select this entry, if the settings are to be checked for all images. This option is preset.
- *Never*: Select this entry, if the settings are never to be checked. With this option, the system will jump over some steps in the analysis and the *Image results* step will be displayed. In general, this setting is only sensible if you have saved the settings to be used as a parameter set and you load them before starting the analysis.
- *First image*: Choose this entry, if the settings are only to be checked for the first image and are then to be used for all other images (even from other samples).
- *First image per sample*: Choose this entry, if you have several samples (with several images per sample) and the settings are to be checked for the first image of each sample.
- *First image per scan area*: You'll only see this entry when you have chosen the *Stage Path* option. Select this entry, if the settings are only to be checked for the first image in each scan area and if the same scan area is to be used for other images.
- *Image interval*: Select this entry, if you would like to analyze several images and would like to check the settings at regular intervals. If this entry is selected, the *Image interval* field will become active. In this field you could, for instance, enter 10 to check the settings for every tenth image.

10265 12062012

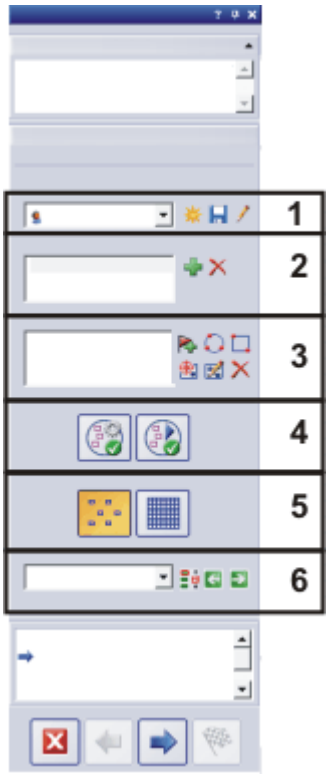
Setting the stage path

The *Materials Solutions* tool window leads you step-by-step through a materials science measurement. In the *Stage path settings* step, you define a stage path on your sample.

What is a stage path?

For most materials science analysis processes, you can define several stage positions on each sample and can save them as a stage path. Here, stage positions can either be entire scan areas or individual XY positions.
The stage path contains the number of samples to be analyzed, and information about which scan areas and/or XY positions are defined on each sample.
For the materials science analysis, the stage will move to the defined positions one after the other. At each XY position, an image will be automatically acquired. For a scan area, several images will automatically be acquired and will be assembled into a single image. Each image acquired will be analyzed with the selected materials science analysis process.

The following stage path settings are available.



(**1**) Choosing a stage path
(**2**) Defining samples
(**3**) Defining scan areas and/or XY-positions
(**4**) Aligning the sample
(**5**) Selecting the inspection mode
(**6**) Selecting the focus mode

(1) Choosing a stage path

To be able to take materials science measurements at different positions on one or more samples, you have to define a stage path. You can use a saved stage path or you can define a new one.

Defining a new stage path

1. Click the *Creates a new stage path* button, to define a new stage path.
   - A stage path is always linked to at least one sample. With the new stage path, a new entry in the *Samples* list will always also be produced. If you click on the *Creates a new stage path* button, the *Sample information* dialog box will be opened first.

2. In the *Sample information* dialog box, you enter information about the sample. By default, the *Reference*, *Group*, and *Comment* fields are available to enter details for the sample.
   - If you have changed the default settings, the *Reference* and *Group* fields can also have another name. You can change the default settings in the *Tools > Options > Materials Solutions* dialog box.
   - You'll see this information when you create a workbook or a report at the end of the analysis.
3. Close the *Sample information* dialog box with *OK* to create the new stage path.
   - The new stage path is added to the *Stage path* list. Once created, the stage path is empty and still has to be completely defined.
   - Now define scan areas and/or XY-positions on your sample.

Note: There can only ever be one stage path active. If you define a new stage path, you will automatically remove all of the currently defined samples and stage positions. You should thus save a stage path which you would like to use again before defining a new stage path.

Saving a stage path

Click the *Saves the current stage path* button, if you would like to use a stage path for several analyses. The following information will be saved:

- The number of samples
- The data entered about the sample
- All of the defined stage positions, i.e. the flags for individual XY-positions and all defined scan areas
- Inspection mode and focus mode

Using an existing stage path

In the *Stage path* list, you will find all of the stage paths that already exist.

1. Select a stage path from the list to load the sample information and stage positions defined in the stage path.
   - If one of the positions in the stage path is outside of the currently defined stage area, you will be presented with an error message. In this case you will not be able to load the stage path.

Note: The *Stage path* list contains the stage paths saved by you as well as those saved by any other user with *Public* access rights. You will not see stage paths saved by other users with *Private* access rights.

You can edit the stage path and thus adapt it to the current sample.

1. Double click on an entry in the *Samples* list to open the *Sample information* dialog box. Here, you can change all of the loaded sample information.
2. Define new stage positions for individual samples, or delete individual stage positions from the *Scan Areas* list.
3. Click this button next to the *Stage path* list to save the altered stage path under a new name or to overwrite the existing stage path.

Managing existing stage paths

Click this button next to the *Stage path* list to open the *Manage Stage Paths* dialog box. Here, you can copy an existing stage path, rename, or delete it.

Note: Public stage paths can be edited, and even deleted by every user of your software.

(2) Defining samples

Prerequisite: The *Samples* list isn't available for all materials science analysis processes.

The *Samples* field lists all samples which are defined in the current stage path. After the name of the sample, in brackets you will find the number of stage positions currently defined for this sample.

Adding and deleting samples

Click this button , to add a new sample to the current stage path. The *Sample information* dialog box automatically opens. Here, you can enter information about the sample.

Select one of the samples listed. Click this button , to delete the selected sample. All scan areas and XY-positions which were defined for this sample will also be deleted.

Viewing and changing the sample data

Double click on a sample to open the *Sample information* dialog box with the current sample information and, if necessary, to edit it.

(3) Defining scan areas and/or XY-positions

Use the *Scan Areas* group to define stage positions on the selected sample, to edit existing stage positions, and to move the XY-microscope stage.

The following buttons are available:

	Adding XY-positions
	Adding scan areas
	Moving the XY stage to the selected stage position
	Editing stage positions
	Deleting stage positions

Adding XY-positions

You can mark several positions on your sample. At each XY-position, an image will be acquired and will be analyzed with the selected materials science procedure.

1. Select a sample from the *Samples* list.
2. Move the XY stage to a position on the sample, at which you would like to take the current materials science measurement.
   - To navigate the XY stage, you can for example use the *Microscope Control* or the *Stage Navigator* tool window. Both tool windows will automatically be displayed in the *Stage path settings* step in the analysis.
   - In the *Stage path settings* step, your system will automatically switch to live mode, so that you can examine the live image to check whether the position on the sample is suitable for analysis.
3. Click this button , located next to the *Scan Areas* list.
   - The current position of the XY stage will now be saved and assigned to the selected sample.
   - The defined XY-position will be marked by a flag in the *Stage Navigator* tool window.

4. Move the XY stage to the next position on the sample, where you would like to take a measurement.
   - The stage will later be moved to the positions specified and in the sequence specified in the *Scan Areas* list. Take this into account when defining the stage positions.
5. Click the button again.
6. Repeat the two last steps until you have defined all of the positions on the sample.

Adding scan areas

Instead of individual positions, you can also define a whole area on your sample for materials science analysis. This area can be rectangular or circular.

1. Click this button to define a rectangular scan area. To do so, you move on the sample, with the motorized XY stage, to the rectangular area's 's top left-hand corner, then to its the bottom right-hand corner.
2. Click this button to define a circular scan area by moving the XY stage. You define the scan area by moving your XY stage to three points, which are on the edge of the round scan area. Your software will help here with corresponding message boxes.
   - Your software will automatically calculate how many individual images are required to completely acquire and analyze the defined sample area. The number of the individual images depends on the current magnification. If you change the magnification, the number of images will be recalculated. You do not have to redefine the scan area.
   - The scan area is displayed in the *Stage Navigator* tool window. In the stage navigator's image display area, you can directly see how many individual images are needed for the defined area at the current objective magnification. If you change the magnification, the display will be updated.
   - The stage will later be moved to the positions specified and in the sequence specified in the *Scan Areas* list. Take this into account when defining the stage positions.
3. In the *Inspection Mode* group, select how the scan areas are to be analyzed.

Editing stage positions

You can redefine scan areas and XY-positions which have already been defined. In contrast to deleting a stage position and then adding a new one, the name of the stage position will not be changed.
You can for instance use this option to adjust an existing stage path for a different sample.

1. From the *Scan Areas* list, select one of the stage positions shown e.g. *Rectangle 2*.
2. Move the XY stage to the position on the sample, to which you would like to move the selected stage position.
3. Click this button , to redefine the selected *Rectangle 2* stage position. For a scan area, you will also have to redefine the size in this case.
   - The name of the new stage position will remain unchanged *Rectangle 2*.

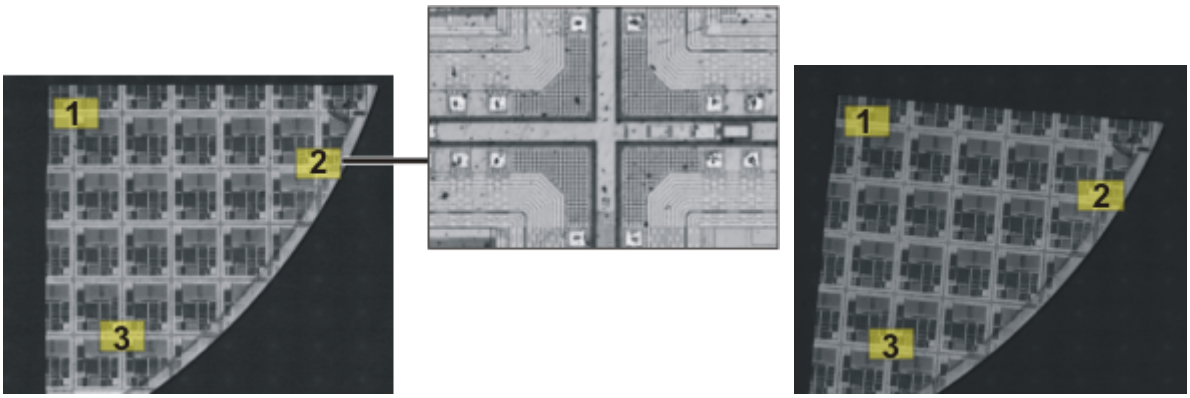
(4) Aligning the sample

With some materials science inspection modes, the measurement has to be carried out at certain positions on the sample. In this case, all samples on the stage have to be positioned the same way so that the stage path can go to the correct positions on the sample. Use the functions in the *Sample alignment* group to compensate for differing alignments of the samples on the stage.

Example: You can use the *Automatic Measurement* solution to measure test structures on a wafer. On the wafer, define three positions that are located on every wafer to be measured. If you now put a new wafer to be measured on the stage, at the beginning of the measurement, move the stage to the three reference positions. This allows your software to recalculate the stage path.

Defining the reference position

1. Click this button to start the definition of the reference position.
   - A yellow triangle on the button indicates that no reference positions have been defined yet for this stage path.
   - The *Acquire Reference Images for Sample Alignment* dialog box opens. It guides you step-by-step through the definition of the reference position.
2. Move the stage to reference position 1 and focus.
   In order for the sample alignment to work well, the reference positions should meet the following conditions.
   - The reference positions should be unambiguous.
   - The reference positions should be as easy to find on the sample as possible.
   - The reference positions should be as far away from each other as possible.
3. Click the *Next >* button to move to the next reference position.
   - Your software now acquires an image at the first reference position. This image is saved as a reference image together with the stage path.
4. Define reference positions 2 and 3.
5. Click the *Finish* button to finalize the definition of the reference positions.
   - The button in the *Sample alignment* group changes its appearance. A green check on the button shows that reference positions have been defined for this stage path.
6. Click this button next to the *Stage path* list, to save the stage path along with the reference positions and the reference images.



On the **left** is an overview of a whole sample. Define three reference positions (1-3) on the sample. A reference image is acquired at each reference position. The illustration shows the reference image at position 2. The reference image is displayed in the live-image during the alignment of the sample to assist you with positioning.
On the **right** is a similar sample that is positioned differently on the stage. The same stage path can be used on both samples with the aid of the reference positions.

Aligning a sample

1. Begin a materials science analysis process that contains a stage path. Reference positions for the stage path are already defined.
   - Your software automatically starts a wizard in the *Define Stage Path* step in the analysis. You can cancel the wizard if you don't want to align the sample yet.
2. Click the *Yes* button in the message box or click the *Align images for sample alignment* button, shown above, to align the current sample with the aid of saved reference images and reference positions.
   - The *Align images for sample alignment* button is only available if reference positions have been defined for the selected stage path.
   - A yellow triangle on the button indicates that the current sample isn't aligned yet.
   - The *Align images for sample alignment* dialog box opens.
3. Decide how the reference image should be displayed. You have the following options in the *Align images for sample alignment* dialog box:
   - Select the *Show reference image as thumbnail* option. Now the reference image for the current position will be displayed as a small image on the top left of the live-image.
   - Select the *Show reference image in overlay* option. Now the reference image is superimposed in full size on the live-image. Use the *Display opacity* slide control to change the transparency of the reference image. The smaller the value, the more transparent the reference image is. Select the value 0 if you don't want to see the reference image for orientation.
4. Move the stage to the required reference positions one after another. Orientate yourself using the displayed reference image.
5. When you've moved to the third reference position, click the *Finish* button.
   - Your software now compares the positions saved in the stage path with the current positions you moved the stage path to and positions the stage path accordingly.

- The button in the *Sample alignment* group changes its appearance. A green check on the button shows that the sample is aligned.

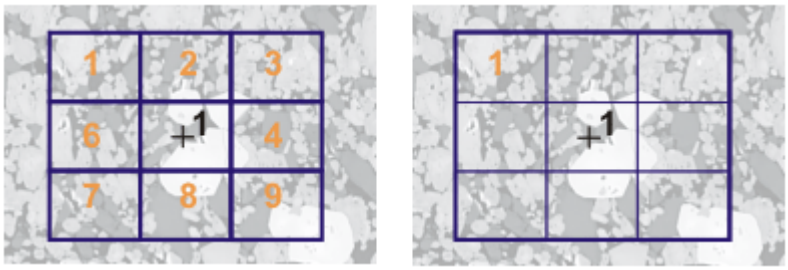
(5) Selecting the inspection mode

Prerequisite: The options in the *Inspection Mode* group are only relevant for scan areas, not for XY positions.

Select the *Single frame inspection* option. Now, all of the images from a scan area will be individually analyzed with the selected materials science method.

Select the *MIA image inspection* option. Now, all of the images acquired from a scan area will be assembled directly as they are acquired, like a puzzle, into a stitched image, to be analyzed with the selected materials science method.

In *MIA inspection mode*, the individual images are acquired with a certain overlap area. Your software will then use pattern recognition to look for two images with the same image information, in the overlap area.
You determine the size of the overlap area in the *Acquisition Settings > Acquisition > Automatic MIA* dialog box. You can open this dialog box, for example, via the *Process Manager* tool window. In the tool window's toolbar, click the *Acquisition Settings* button. Select the *Acquisition > Automatic MIA* option in the tree view.



The illustration shows a sample on which one scan area (1) is defined. 9 individual images are needed to fully acquire the scan area.
On the left, the *Single frame inspection* option is selected. If, for example, you do a phase analysis and output a workbook as a result, you will now find the results for 9 images on the sample's worksheet.
On the right, the *MIA image inspection* option is selected. On the sample's worksheet you will now find only one result for the same scan area, as the individual images will be assembled to a single image before the analysis.

(5) Selecting the focus mode

If you use a stage path, during the measurement, the stage will move to various positions which can be far removed from one another. In this case it will generally be necessary to refocus several times during the measurement, so that each individual image is ideally focused and can be analyzed.

From the *Focus Mode* list, select one of the following options:

- Not refocusing on samples
- Manually refocusing on samples
- Using a focus map
- Using the software autofocus

The selected focus mode applies for the entire stage path, which means for all samples and all stage positions.

10801 23052013

7.2. Chart comparison

What are chart comparisons?

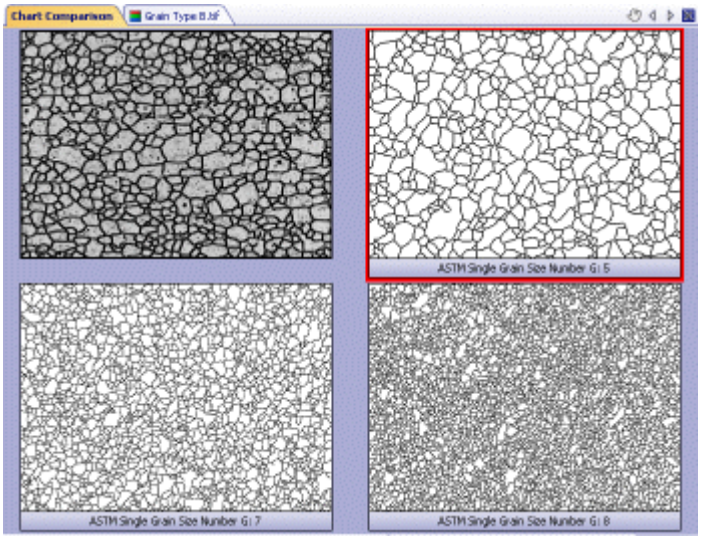
In metallography, chart comparisons are used as a means of quality control. They make it possible to compare an image with numerous reference images. The reference images are a part of the industry standards (which have to be purchased) by which the chart comparisons are carried out.

Example 1:
During a qualitative grain size analysis, you determine the grain size of metallic samples. You compare the images that are to be checked with the reference images. You assign the reference image with grains of the same size to each of the images that is to be checked.

Example 2:
During a quality control, you check various components to see if they are free of defects. To do so, you compare the components with images of various components that are either defective or free of defects. You assign the appropriate reference image to the object that is to be checked.

General procedure for a chart comparison

The image that is to be checked, and all or some of the reference images, are displayed simultaneously on the screen. Your software makes sure that all of the images are always shown on the same scale. By making a visual comparison, the user finds out which of the reference images is the most similar to the image that is to be checked. Saved along with every reference image is the value that it was assigned by the industry standard. By the selection of a reference image, the image that is to be checked is assigned this value, too.



The above image shows the document group during a chart comparison. The image that is to be checked is located at the top left, the reference images are arranged either next to it or beneath it. The selected reference image is framed in red.

Results

The results of a chart comparison can be output in a workbook. As well as that, when you carry out chart comparisons on live-images, you can immediately reject the samples that don't meet the required values.

If the Chart Comparison analysis process isn't displayed in the Materials Solutions tool window

To be able to carry out chart comparisons with the image analysis program, the charts from at least one industry standard have to be installed. Only then will the *Chart Comparison* analysis process be displayed in the *Materials Solutions* tool window. The industry standards that are to be used for the chart comparison have to be purchased. They can be purchased through Olympus Soft Imaging Solutions. You will receive a DVD for each industry standard that you purchase.

Use the Quick Setup Guide which accompanies the DVD to install the industry standard's charts.

Note: Even if you haven't purchased an industry standard yet, you can still view the *Chart Comparison* analysis process. To do this, install a demo plate. Using this, you can get an impression of how this analysis process works. Real analyses (complying with industry standards) are, however, not possible using these demo plates. An instruction on how to install demo plates can be found in the online help.

00723 06052013

Performing a chart comparison

Prerequisites:

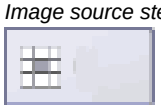
The *Chart Comparison* analysis process is only displayed in the *Materials Solutions* tool window when you have purchased at least one industry standard and have installed its' charts.

Even if you haven't purchased an industry standard yet, you can still view the *Chart Comparison* analysis process. To do this, install a demo plate. You can carry these step-by-step instructions out with the *Demo single grain size* demo plate. An instruction on how to install demo plates can be found in the online help.

Note: Real analyses (complying with industry standards) are, however, not possible using these demo plates.

Example image FerriteGrains.tif
When your software was installed, several example images have been copied automatically. You can follow these step-by-step instructions when you use the example image FerriteGrains.tif. Open this image and make sure that it has been selected in the document group. You can find the information where the example images are located in the online help.

Image source step



1. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
2. Click the *Chart Comparison* button.
   - As soon as you've started this analysis process you'll be guided step by step through the measurement. A lot of your software's other functions will not be available while an analysis process is running.
   - The *Materials Solutions* tool window displays the *Image Source* step.
3. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
4. Select the *Skip 'Sample information'* check box if you don't want to add any details about the sample or about an image of the sample. If you want to add details, make sure the check box is not selected.

Note: If you want to analyze images from more than one sample in the same analysis process, the *Skip 'Sample information'* check box must be cleared. Only then will the *New Sample* button be displayed. With this button, you can specify when an image to be analyzed belongs to a new sample.

5. Select the *All images* entry in the *Check settings and results* list.
   - If you would like to analyze your own images later on, you can also select another entry from this list, e.g. if you would no longer like to check the settings for every image.
6. Click the *Next* button.

- The *Materials Solutions* tool window will display the next step.

Sample information step

Note: You will only see this step in the analysis if, in the previous step, the *Skip 'Sample information'* check box wasn't selected.

1. Enter information on your sample. By default, these fields are called *Reference* and *Group*.
2. If you want to, enter a comment about the sample. This comment is valid for all of the images of this sample.
3. If you want to, enter a comment about the current image, too.
4. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

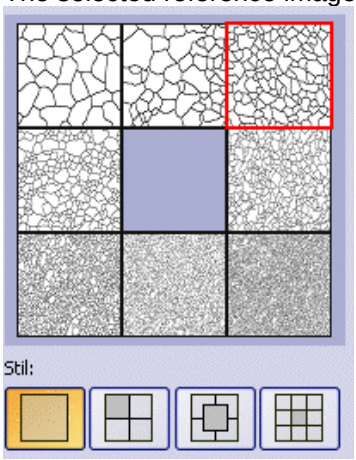
Settings step

1. Select the chart by which you want to analyze the image. If you've installed a demo plate, select that.
   - For the FerriteGrains.tif image, you can select the *Single grain size* entry in these step-by-step instructions to specify the grain size. You'll only see this entry if you've chosen the *Demo single grain size* demo plate.
2. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.
   - In the document group, the new *Chart Comparison* document will be displayed.

Comparison step

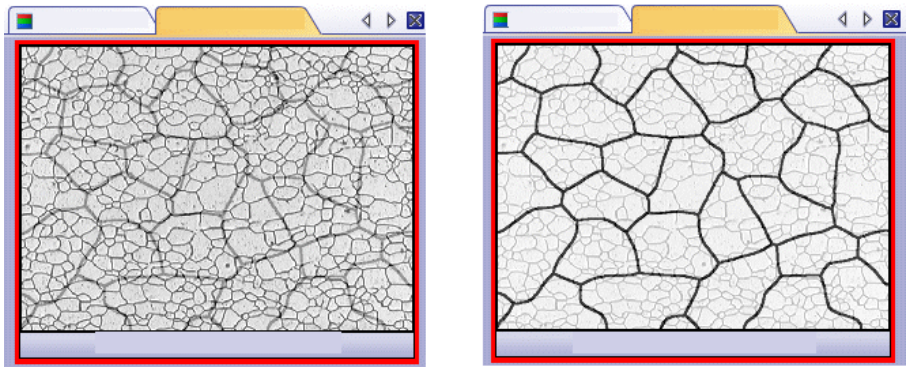
1. In the *Style* group, choose how the images for the chart comparison are to be arranged in the document group. Choose an arrangement in which the FerriteGrains.tif image and the selected reference image are superimposed.

   To do this, click this button.
   - In the document group, the *Chart Comparison* document will now be displayed. It contains exactly one image.
   - In the *Overview* field, you see the arrangement that has been chosen. The selected reference image is framed in red.

2. Compare the structures of the current image with those of the reference image. Move the slide control below the *Style* field towards the *Opaque* position, if the image that is to be checked is to superimpose the reference image. Alternatively, move the slide control towards the *Transparent* position,

if the image that is to be checked is to be superimposed by the reference image.



The illustration on the left shows the image that is to be checked. Because the slide control is located in near the *Opaque* position, the reference image's structures can only be faintly recognized. For the illustration on the right, the slide control has been moved towards the *Transparent* position. Now, the reference image can be clearly recognized, and the image that is to be checked can be only faintly recognized.

3. If you want to choose another reference image, in the *Comparison* group, click that image with your left mouse button.
4. When the reference image that is the most similar to the image that is to be checked, has been chosen: Click the *Accept* button.
   - The chosen image's data will be accepted in the *Results* field.
   - It's possible to accept several reference images, for example, with samples that have very different structures.
5. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Note: When you carry out analyses on the live-image: Click the *Get Results* button. You will then see the *Results* step. Otherwise, when you've finished analyzing one live-image, the next live-image will always then automatically be offered for analysis.

Results step

1. Select the *Generate Workbook* check box, to have a document of the "workbook" type automatically created at the end of the analysis.
2. Click the *Finish* button.
   - The *Materials Solutions* tool window switches back to the start position. You can now use all of your software's functions again.

00724 15052013

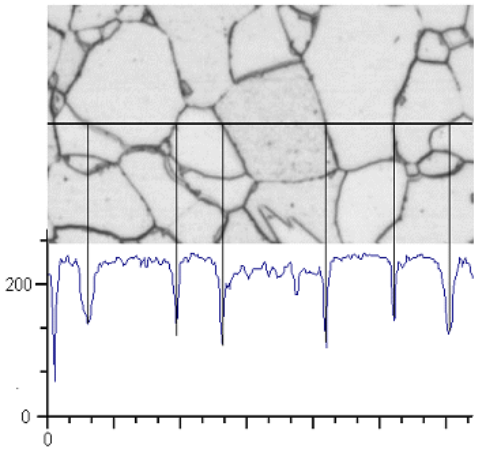
7.3. Grains Intercept

What is an intercept analysis?

The intercept analysis is used to measure grain sizes and to document them. It is often used in material analyses, for example, when the quality of steel or other metals is being tested.

When an intercept analysis is made, measuring lines are placed in an image. Along these measuring lines, your software searches for abrupt deviations in the pixels' intensity (gray value). An intensity deviation occurs, for example, if dark pixels are present in an image made up of mainly light pixels. When an intensity deviation exceeds the parameters that have been set, an intercept point will be plotted at this position on the measuring line.

The intercept points are counted. The distance between two intercept points is also measured. From this measurement, the mean intercept length is calculated.



Description of the above illustration

The intensity profile is determined along the horizontal measuring line. Whenever the measuring line crosses a grain boundary, this leads to a distinctive minimum in the intensity profile. When an intercept analysis is made, these minima in the profile are used to determine the intercept points. In the illustration shown, the grain boundaries are dark, the process can, however, also be used on images with light grain boundaries. The analysis of cascaded grain boundaries (with multi-phase materials) is also possible.

Results of an intercept analysis

An intercept analysis provides the so-called G-value, which is defined as a characteristic grain size in the corresponding industry standards. G is calculated from the number of intercept points and the mean intercept length. The grain sizes are measured in accordance with the industry standards:

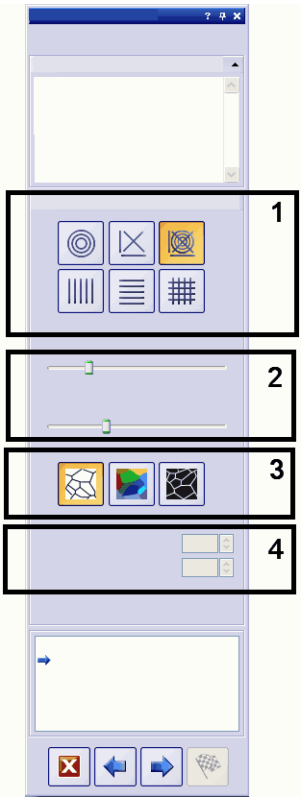
- ASTM E112
- GB/T 6394
- GOST 5639
- ISO 643
- DIN 50601
- JIS G 0551
- JIS G 0552

The results of an analysis can be displayed in a workbook. Additionally, or alternatively to that, the results can be displayed in an MS-Word report.

00700

Settings for the intercept analysis

In this step, you make important settings for the analysis. The following options are available:



(1) Selection of the line pattern

The line pattern determines along which lines the intercept points are looked for. At every position along the line, intensity deviations will be searched for in the intensity profile. As soon as an intensity deviation fulfills the definition criteria set, it will be displayed as an intercept point in the image. Which line pattern is suitable for a specific task, depends on the type of structures that are to be measured, and their position in the image.

The following line patterns are available:

Circles Three circles are placed in the center of the image. The size of the measurement pattern corresponds to the diameter of the largest circle. This line pattern is appropriate for images with structures distributed equally throughout the image or structures which progress from the middle of the image to the edges.

Cross The cross consists of two diagonally crossed lines, as well as a line each below and to the left of this cross. The size of the measurement pattern corresponds to the length of the horizontal line below the cross.

Cross and Circles The *Cross and Circles* line pattern combines the two line patterns *Cross* and *Circles*.

Horizontal Lines With this line pattern, horizontal lines are distributed evenly across the measurement pattern.

Vertical Lines With this line pattern, vertical lines are distributed evenly across the measurement pattern.

Horizontal and vertical lines With this line pattern, horizontal and vertical lines are distributed evenly across the measurement pattern, forming a grid.

(2) Slide controls for changing the results displayed

Two slide controls are available. You can change the position of the slide controls however you want to in this step. This has an effect on the number of intercept points that will be found. Therefore you should keep an eye on the display in the image.

Grain boundary width

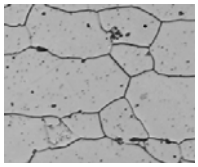
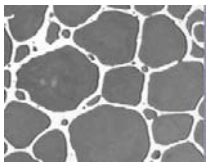
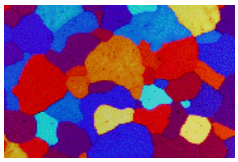
Here, you set the necessary width for the detection of a grain boundary. When a small grain boundary width is set, your software finds considerably more intercept points than with a wider grain boundary.

Noise reduction

Use this slide control to apply a smoothing filter to the image. The smoothing filter reduces the image noise. You should therefore apply a smoothing filter to images that are very noisy before the intercept analysis is made. Move the slide control from the left to the right, to increase the strength of the smoothing filter in small steps. This will lead to a reduction of the detected intercept points.

(3) Buttons for selecting the grain boundary type

Here, you specify which criteria are used to detect the grain boundaries. Depending on the image that is to be analyzed, the grain boundary type can be dark (left illustration) or light (middle illustration). Where images that don't have any intensity deviations, but only show different gray values, are concerned, select the *Step* setting (right illustration).

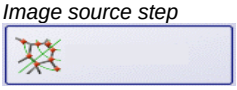
(4) Number of test lines

These fields are only active if you selected a line pattern that contains horizontal or vertical lines. In this case, you specify here the number of lines to be used for the intercept analysis.

10263

Performing an intercept analysis

Image source step



1. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
2. Click the *Grains Intercept* button.
3. In the *Image source* group, choose the image or the images that you want to analyze. When you do this, pay attention to the information as to how many images have been selected This information is shown in bold font at the bottom of the group. The following options are available:
4. Decide whether you want to load settings that you have saved while you were analyzing another image. Then you can, if necessary, adapt these settings and apply them to this image. Click the *Load from file...* button to load the settings that have been saved.
5. Decide whether or not you want to add data about the sample or about individual images while the analysis process is in progress. If you don't want to do so, select the *Skip 'Sample information'* check box.
   Should you want to add data, (e.g., because you are analyzing images of several samples in the same analysis), leave the check box unselected.
6. Select the *All images* entry in the *Check settings and results* list.

- If you would like to analyze your own images later on, you can also select another entry from this list, e.g. if you would no longer like to check the settings for every image.

7. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.
   - Should you be analyzing the live-image and a database is open, you'll be asked whether you want to save the acquired individual image in the database.

Sample information step

Note: You will only see this step in the analysis if, in the previous step, the *Skip 'Sample information'* check box wasn't selected.

1. Enter information on your sample. By default, these fields are called *Reference* and *Group*.
   - If you have changed the default settings, these fields can also have another name. Additional information on changing the default settings can be found in the online help.
2. If you want to, enter a comment about the sample. This comment is valid for all of the images of this sample.
3. If you want to, enter a comment about the current image, too.
4. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Settings step

1. Choose a line pattern that is appropriate for the structures in the image that is to be analyzed. You can choose between various line patterns.
   - The pattern determines along which lines, intercept points in the image are looked for.
2. Take a look at the intercept points that have been found in the image. If necessary, change the settings to optimize the results shown.
3. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Image results step

1. Check the results shown. You can see the results of the current image, and the overall results of all of the images that have already been analyzed for this sample.
2. Should you not be satisfied with the results for the current image: Click the *Back* button, to switch back to the *Settings* step. Then you can try to improve the results for this image by choosing another line type or by moving the slide controls to another position.
3. Should you want to correct the intercept points that have been automatically found, click the *Add Intercepts...* or *Delete Intercepts...* buttons. This will enable you to add intercept points manually, or to delete superfluous intercept points.
   - You can find step-by-step instructions on how to correct intercept points in the online help.
4. Select the *Check settings* check box to have the *Settings* step displayed with every image.
   - This enables you to individually change the slide control's setting before every individual image is analyzed. This makes sense, for example, if the images that are to be analyzed are of greatly different quality, and you need to adjust the noise reduction individually for each image.

- The *Check settings* button is only active if you analyze several images of a sample at the same time.

5. When you analyze images that you selected before the analysis began: Click the *Next* button.
   - Should you analyze images from the database, you will then be asked whether you want to save the changed images, or not. You can either insert the analyzed images as new images into the database, or overwrite the existing database images with them. As well as that, you can either save the images in the file system or reject them.
   - The *Materials Solutions* tool window will display the next step.
   - Only when you carry out an analysis on the live-image, or you want to leave out the analysis of all of the remaining images: Click the *Get Results* button. You will then see the *Results* step. Otherwise, when you've finished analyzing one live-image, the next live-image will always then automatically be offered for analysis.

Results step

1. Check the results shown. You can see the overall results for all of the images, that have already been analyzed for this sample.
2. Select the *Generate Report* check box, if you would like to have a report automatically generated in MS-Word once the analysis is completed.
   - The additional step *Reporting* will be added to the current analysis. In the lower part of the dialog box, the *Finish* button will change into the *Next* button.
3. Select the *Generate Workbook* check box, to have a document of the "workbook" type automatically created at the end of the analysis.
4. If you want to save the current settings to a file, click the *Save settings....* button. Then assign a descriptive name in the next dialog box.
   - You can load these settings (parameters) when you analyze further images. To do so you must load the image to be analyzed and, in the *Image source* step, click the *Load from file...* button. The sample and image comments, the line pattern used, and the position of the slide controls in the *Settings* step will be saved.
5. Click the *Next* button.

Reporting step

1. Select the *Default* option to use the document template that has been defined as the default document template. The document template determines, e.g. the appearance of the report's header and footer.
   - Should you want to change the default document template, use the *Tools > Options > Report Composer > Document Templates* command. Add the document template you want to the *Templates* list, select it, and click the *Set as Default* button.
2. In the *Content* group, select the check box for the pages the report should contain.
   - Select the *Summary page* check box, if the first page of the report is to contain a summary of all of the results of the current analysis. The creation of a summary page can for example be useful, when you have analyzed a large number of images that belong to different samples.
   - Select the *One page per sample* check box, if the report should contain one page for every sample. This page displays the overall results for all of the images belonging to that sample. Using this setting is a good idea, for example, when you have analyzed images of different samples.

- Select the *One page per image* check box, if the report should contain a page of its own for every image. Should only this check box have been selected, and you have analyzed three images, your report will contain exactly three pages.
- Select the *Show results in overlay* check box if the image layer that contains the results is to be displayed along with the images.

3. Click the *Finish* button.
   - The report will be generated and displayed in MS-Word.
   - The workbook will be created. It always contains a minimum of two worksheets. On the first worksheet, you'll see a summary of the results. On the second worksheet you'll see the details concerning the sample used. Should you have analyzed several samples, the workbook will contain additional worksheets.
   - The *Materials Solutions* tool window switches back to the start position. You can now use all of your software's functions again.

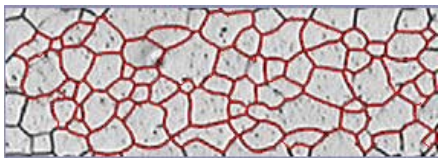
00701 12062012

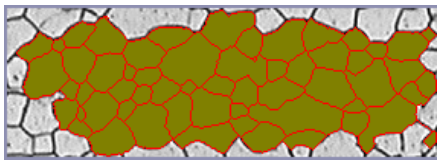
7.4. Grains Planimetric

What is Grains Planimetric?

The grains planimetric analysis is used to measure grain sizes and to document them. It is often used in material analyses, for example, when the quality of steel or other metals is being tested. The grains planimetric analysis determines the grain size by means of the grains' area. In this way, it differs from the intercept analysis, that determines the grain size by means of the number of intercept points.

Samples with dark grain boundaries or samples with bright grain boundaries can be used. The analysis of cascaded grain boundaries (with multi-phase materials) is also possible.





The image shown above shows the results of an automatic detection of the grain boundaries. The grain boundaries that have been detected are plotted in red (first illustration). Additionally, it's possible to have the grains that have been found displayed in color (second illustration). When this is done, the original image isn't changed, because this information is written to another image layer.

Editing grain boundaries

You can manually edit the grain boundaries that your software found automatically. When you do this, you have the possibility of deleting superfluous grain boundaries and adding boundaries that are missing.

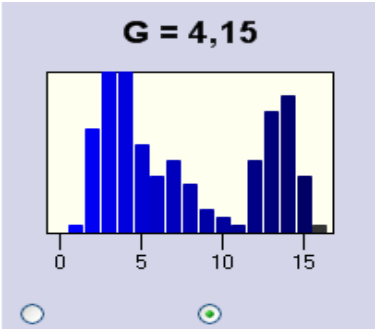
The results of a grains planimetric analysis

A grains planimetric analysis provides the so-called G-value, which is defined as a characteristic grain size in the corresponding industry standards. The grain sizes are measured in accordance with the industry standards:

- ASTM E112
- GB/T 6394
- GOST 5639
- ISO 643
- DIN 50601
- JIS G 0551
- JIS G 0552

Documenting the results

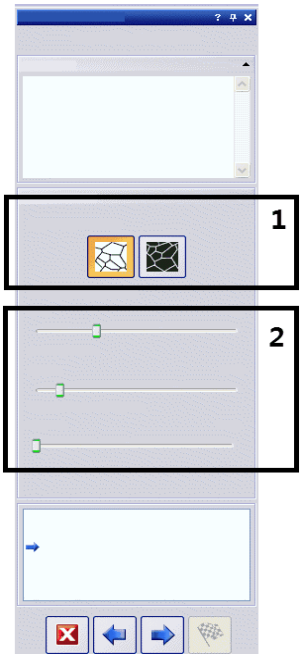
The results of an analysis can be displayed in a workbook and in a chart. Additionally, the results can be displayed in a MS-Word report.



00720

Settings for the planimetric analysis

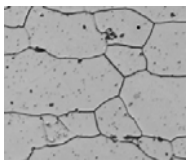
In this step, you make important settings for the analysis. You'll only see some of the setting options described below. Which of them you see depends on the image type you chose in the previous *Image type information* step.

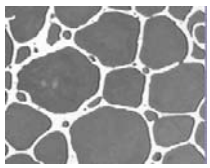


(1) Buttons for selecting the grain boundary type

Prerequisite: You'll only see these buttons if you chose the "Flat etched grains" or "Ferritic grain size with pearlite" image type in the previous step.

Here, you specify which criteria are used to detect the grain boundaries. Depending on the image that is to be analyzed, the grain boundary type can be bright or dark. Therefore, this slide control can only stand at the position *Dark* or *Bright*.





In the illustration on the left, the grain boundaries are dark. In the illustration on the right, the grain boundaries are bright.

(2) Slide controls

The positioning of the slide controls influences the detection of the grain boundaries. While you are positioning the slide controls, observe which grain boundaries are found. The preview is updated after every change in the settings.

Position the slide controls in such a way that the grain boundaries are detected as completely as possible. It doesn't matter if the grain boundaries are interrupted somewhere in between. The algorithm that calculates the G-value will automatically close small interruptions in the boundaries.

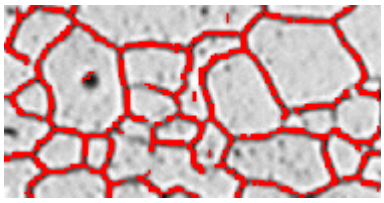
Note: Make sure that the correct grain boundary type has been set, before you adjust the positions of the other slide controls.

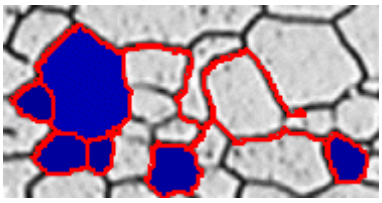
Note: If you are not sure whether or not a slide control is positioned correctly, click the *Next* button and have a look at the results in the *Image results* step. With the *Back* button, you can always return to the *Settings* step.

Grain boundary width

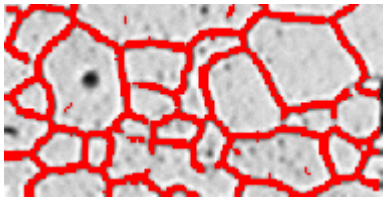
Prerequisite: You'll only see this slide control if you chose the "Flat etched grains" image type in the previous step.

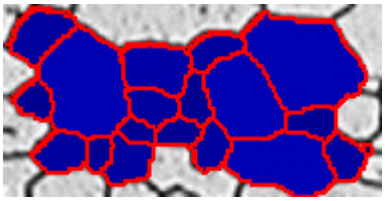
With this slide control, you set the necessary width for the detection of a grain boundary. If the grain boundaries can be clearly recognized, a small grain boundary width is sufficient. The slide control can, in this case, stand on the far left (near the *Small* position). If the grain boundaries are difficult to recognize, you'll need to increase the grain border width. To do so, move the slide control to the right (towards the *Large* position).





In the first illustration, the selected grain boundary width is too small. In the *Image results* step, you can see that the grain boundaries have not been detected completely.





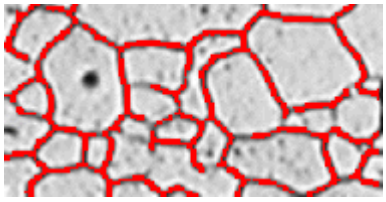
In this illustration, the grain boundary width has been increased. Now the grain boundaries have been completely detected. In the *Image results* step, you can see that the grain boundaries have been detected completely.

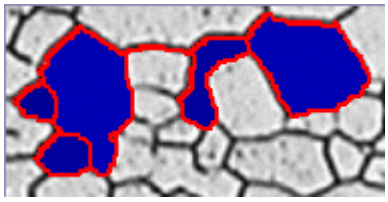
You can use the program options to change the color in which the detected grain boundaries are shown. You can find more information on this topic in the online help.

Threshold

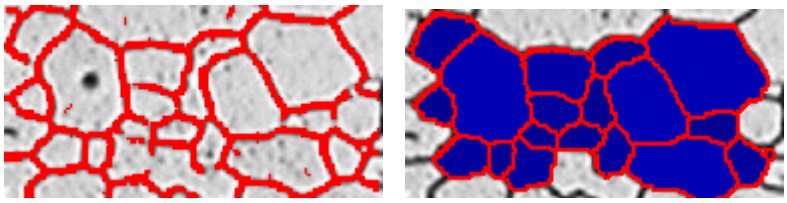
Choose whether a smaller intensity range is sufficient for the detection of a grain boundary. This is, the case, when all of the grain boundaries stand out clearly against the background, for example. In this case, you can move the slide control to the far right (towards the *High* position).

If not all of the grain boundaries stand out clearly against the background, e.g., because some grain boundaries are brighter than others, a larger intensity range has to be defined for the detection of the grain boundaries. In this case, move the slide control to the far left (towards the *Low* position).





In the first illustration, the selected threshold value is too high. In the *Image results* step, you can see that not all of the grain boundaries have been detected.



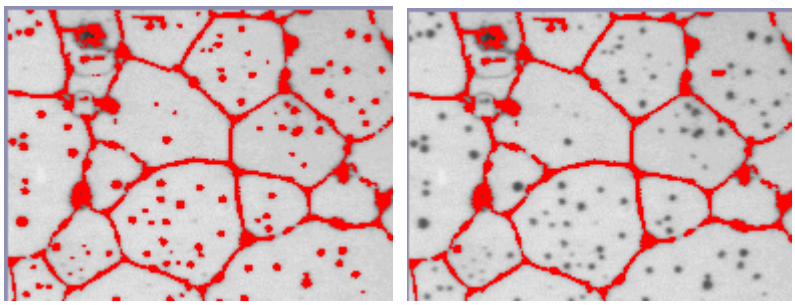
In this illustration, a lower value for the threshold values has been given. In the *Image results* step, you can see that all of the grain boundaries have now been detected.

Artifact removal

With the help of this slide control, you can specify to which size artifacts that are located within the grains are to be ignored for reconstructing the grain boundaries.

Your software now searches for groups of pixels that have the same intensity as the grain boundaries, but which stand alone (i.e. not near a grain boundary). The detected artifacts will be ignored during the next step, reconstructing the grain boundaries. If the slide control is at the *Weak* position, only bigger artifacts will be ignored, while small artifacts will still be detected (see illustration below). If the slide control is at the *Strong* position, even small artifacts will be ignored.

With samples that have a lot of artifacts, using the slide control, you can considerably reduce the time your software needs for reconstructing the grain boundaries.



In the first illustration, no artifacts are ignored (the slide control is at the *Weak* position). In the second illustration, many artifacts are ignored (the slide control is at the *Strong* position). With this setting, the next step, reconstructing the grain boundaries, can be completed much quicker.

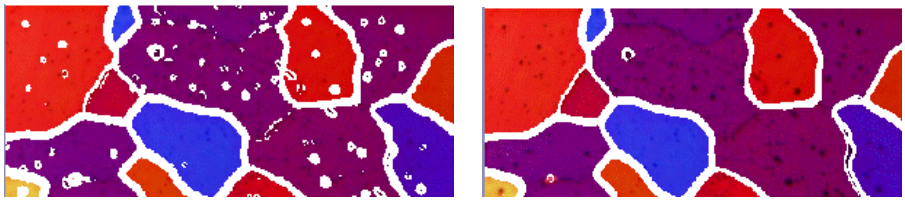
Note: Although the *Artifact removal* slide control's main function is to make reconstructing the grain boundaries quicker, with some samples, it may also lead to better results. This is the case if a sample has so many artifacts lying close to each other that your software makes a new grain out of it. This fake grain would then distort the number of the detected grains and also the G value.

Smoothness

Prerequisite: You'll only see this slide control if you chose the "Color etched grains" image type in the previous step.

With the help of this slide control you can specify that small structures or patterns that are located within the grains are to be ignored for the analysis. These structures have nothing to do with grains. Therefore, it is important to exclude them from the detection. If this is not done, these small structures are taken for grains and will thus affect the result of a planimetric measurement negatively.

Set the smoothness as exactly as possible, so that small structures or patterns only just stop being detected. Don't choose a larger value than necessary. If the image smoothness chosen is unnecessarily great, real small grains won't be detected.



In the first illustration, the selected image smoothness is too small. With this setting, numerous structures (e.g., patterns) within the grains are detected, and this negatively affects the results of the planimetric measurement. In the second illustration, a higher value for the image smoothness has been chosen. You can clearly see that only a few structures were still detected within the grains. Therefore, the result of the planimetric measurement is more exact.

10284 28062012

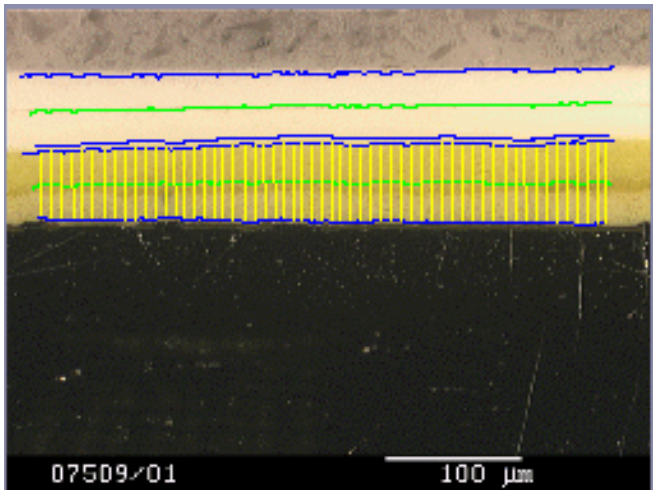
7.5. Layer Thickness

What are layer thickness measurements?

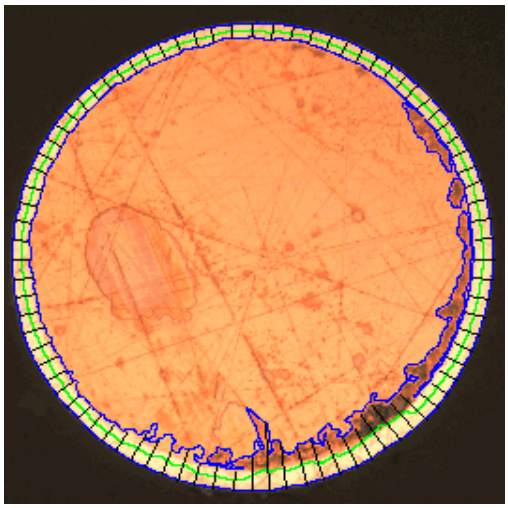
By using layer thickness measurements you can measure layers on calibrated images automatically or interactively. The object that is to be measured is the thickness of one layer or of several layers.

Each layer is defined by two borders and a neutral fiber. The neutral fiber is a reference line which is there to specify the layer's course. The neutral fiber is automatically defined by the program.

You can define either open or closed layer types. When you have a closed layer type, you can measure circular layer structures. In this mode, the measurement line's first point is automatically connected to its last point.



Measuring an open layer: In the image, two layers have been measured. You can see 4 layer borders (blue lines) and two neutral fibers (green lines). The measurement lines (yellow lines) are shown for the currently selected layer.



Measuring a closed layer: In the image, the outer layer has been measured. You can see the layer borders (blue lines), the neutral fiber (green line) and the measurement lines (black lines).

Results of a layer thickness measurement

The results of an analysis can be displayed in a workbook. Additionally, or alternatively to that, the results can be displayed in an MS-Word report.

The borders that have been found, the neutral fibers and the measurement lines will be saved together with the image, if you save it in TIF or VSI format. This information will be saved in a separate image layer that you can show and hide via the *Layers* tool window.

General procedure for a layer thickness measurement

1. Select the analysis process

Click the *Layer Thickness* button, located in the *Materials Solutions* tool window.

▼

2. "Image source" step

Choose the image you want to measure.

▼

3. "Settings" step

Choose the definition method you want to use (Automatic, Manual, Magic Wand).

▼

4. "Magic wand" step

If you have selected the *Magic wand* definition method: Define the contours.

▼

5. "Define borders" step

If you have selected the *Automatic* or *Magic wand* definition method: Define the borders.

▼

6. "Define layers" step

Define the layers.

▼

7. "Image results" step

Take a look at the measurement results.

▼

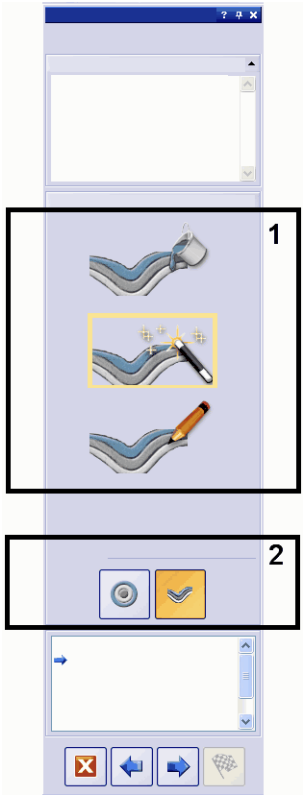
8. "Results" step

Document the results (report or workbook).

00725 12062012

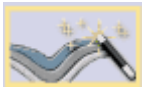
Settings for layer thickness measurements

In this step, the following possibilities are available:



(1) "Settings" group

In the *Settings* group, choose how the contours are to be defined. To do this, click the corresponding icon. You can choose between the following definition methods.

- an automatic definition
- a manual definition
-  a definition with the magic wand

The current definition method is outlined in yellow.

An **Automatic definition** is suitable for samples whose layers feature distinct intensity differences (e.g., light layers in front of a dark background). With these samples, as a rule, the automatic threshold value setting used for this definition method functions well.

A **Definition by Magic wand** is suitable for samples that have irregular borders, that would be very difficult to trace manually.

A **Manual definition** is suitable for samples in which there are only very small intensity differences, which means that the automatic definition would not provide you with satisfactory results. Also when only a small part of a layer interests you, you can easily set it with the manual definition.

Please note: You can change the definition method during a measurement: For example, you can first have a contour determined by using the magic wand, then add an additional border manually.

(2) "Layer type" group

In the Layer type group, you choose whether open or closed layers are to be defined. To do this, click the corresponding icon.

With an open layer type, you can, e.g., measure layer structures that continue all through the image. When you have a closed layer type, you can measure circular layer structures. In this mode, the measurement line's first point is automatically connected to its last point.

Please note: The *Layer type* can only be specified at the beginning of a measurement. In contrast to the definition method, the *Layer type* can't be changed during the measurement.

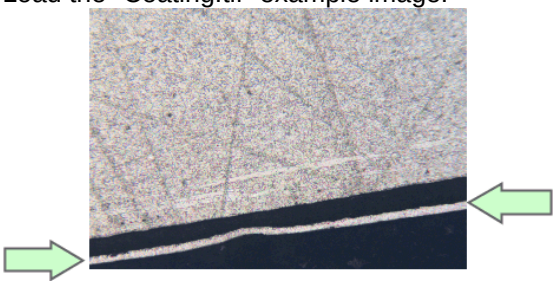
10500

Performing an automatic layer thickness measurement

Note: You can follow these step by step instructions on your PC. They describe a layer thickness measurement on an example image.

Image source step

1. Load the "Coating.tif" example image.



   - On this image, the thin light layer is to be measured.
2. Activate the *Materials Solutions* tool window.

3. Click the *Layer Thickness* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
6. Select the *All images* entry in the *Check settings and results* list.
7. Click the *Next* button.

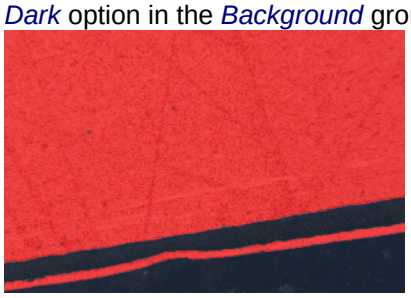
Settings step

1. Click the *Automatic* button.
2. In the *Layer type* group, click the icon for an open layer .
3. Click the *Next* button.

Automatic step

1. You see the image on which some of the image structures are now shown in color, because the first phase was automatically set up.

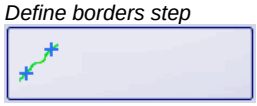
2. Since the required image structures are not yet shown in color, select the *Dark* option in the *Background* group.



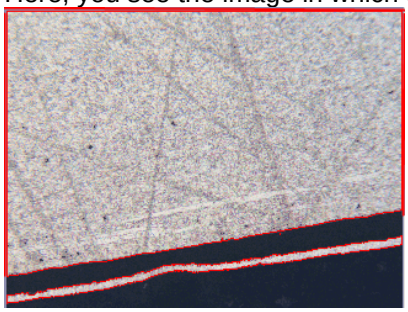
   - Now, the required image structures are shown in color.

3. Click the *Next* button.

Define borders step

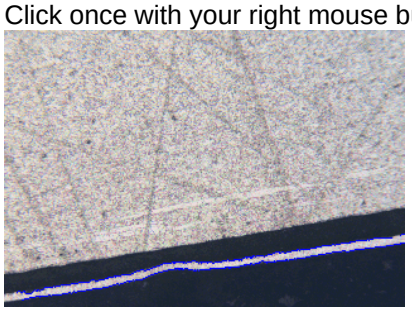


1. Here, you see the image in which the contours are outlined in red.



2. Click the *Define borders...* button.
3. Now, specify which part of the contour represents a border. Click the contour once with your left mouse button, to activate the mode.
   Then click with your left mouse button at the position in the contour where the first border is to begin.
   Then click with your left mouse button at the position in the contour where the first border is to end.
   - The beginning and the end of this border will be indicated by two green crosses.
4. Now, define the second border. To do so, click with your left mouse button again at the position where this border is to begin. Then click with your left mouse button again at the position where this border is to end.
   - The beginning and the end of this second border will be indicated by two blue crosses.

5. Click once with your right mouse button in the image.



- The borders that have been defined will be plotted in blue.

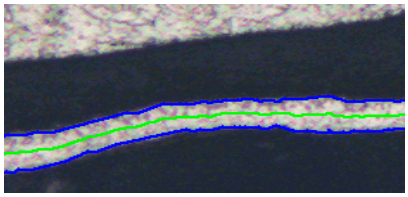
6. Since you don't want to define any additional borders: Then click once more with your right mouse button in the image, to switch off the mode for defining the borders.
7. Click the *Next* button.

Edit borders step

1. Since you have already defined both of the borders, and don't want to change them: Click the *Next* button.

Define layers step

1. Click the *Add layers...* button.
2. Click the first border.
3. Click the second border.

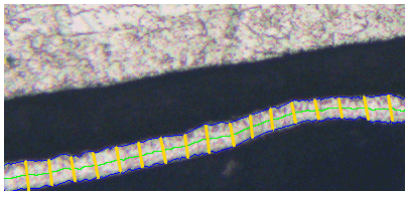


- The layer has now been defined. The neutral fiber is plotted in green. It always lies in the middle of the layer.

4. Click your right mouse button to finish the definition of the layer.
5. Click the *Next* button.

Image results step

1. Take a look at the results of the current image, shown in the *Image results* group. This group contains a table with the measurement results.
   - The values in the *Steps*, *Distance*, and *Type* fields can be edited when you double click in the cell you want to edit. You can find more information on this topic in the online help.
   - The lower part of the group contains several buttons, with which you can change the way the layer thickness measurement is displayed. You can find more information on this topic in the online help.
2. Check the results shown in the image.



- The measurement lines are shown in yellow in the image.

3. Click the *Next* button.

Results step

1. Select the *Generate Report* check box, if you would like to have a report automatically generated in MS-Word once the analysis is completed.
2. Select the *Generate Workbook* check box, to have a document of the "workbook" type automatically created at the end of the analysis.

Reporting step

Define the report that contains the measurement results.

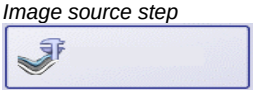
- The *Materials Solutions* tool window switches back to the start position. You can now use all of your software's functions again.
- Through the materials analysis measurement, the image has collected one or more additional layers (can be seen in the *Layers* tool window). If required, save the image in TIF or VSI format to retain these newly created image layers.

00732 12062012

Performing a layer thickness measurement with the magic wand (closed layer)

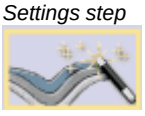
You can follow these step by step instructions on your PC. They describe a layer thickness measurement on an example image.

Image source step



1. Load the "Copper Wire Section.tif" example image.
   - The image shows a cross section through a copper wire. The outermost layer is to be measured.
2. Activate the *Materials Solutions* tool window.
3. Click the *Layer Thickness* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
6. Select the *All images* entry in the *Check settings and results* list.
7. Click the *Next* button.

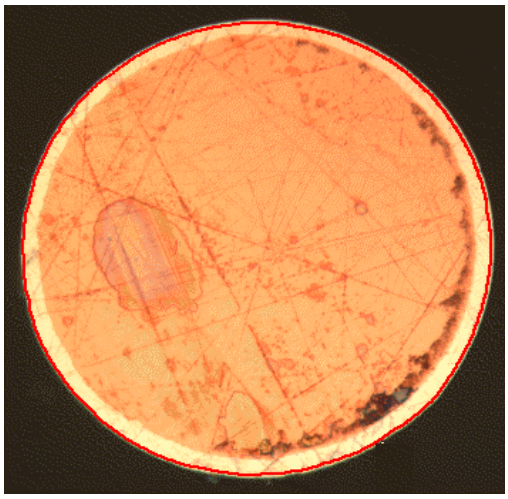
Settings step



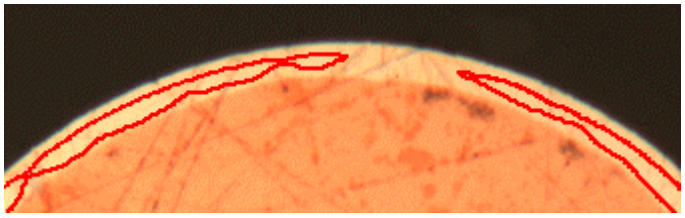
1. Click the *Magic Wand* button.
2. In the *Layer type* group, click the icon for a closed layer .
3. Click the *Next* button.

Magic wand step

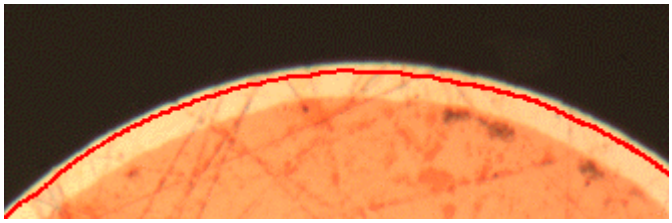
1. Click the *Add contours...* button.
2. Click the button for the *HSV* color space .
3. Then define the first contour. To do this, click once with your left mouse button on a position in the image that lies within the outermost layer.
   - The contour will be shown by a red line.



Note: Make sure that the contour completely includes the outer layer. And that the contour's outline isn't discontinued at any point on the outer layer. Change the position of the slide control in the *Tolerance* field, until the contour completely includes the layer that is to be measured.



Wrong: The contour's outline is noncontinuous.



Right: The contour completely contains the layer that is to be measured.

4. Click your right mouse button to finish the definition of the contour.
   - Now, the first border has been defined. It will be plotted in blue.
5. Click the *Next* button.
   - The *Edit borders* step in the analysis will be shown.

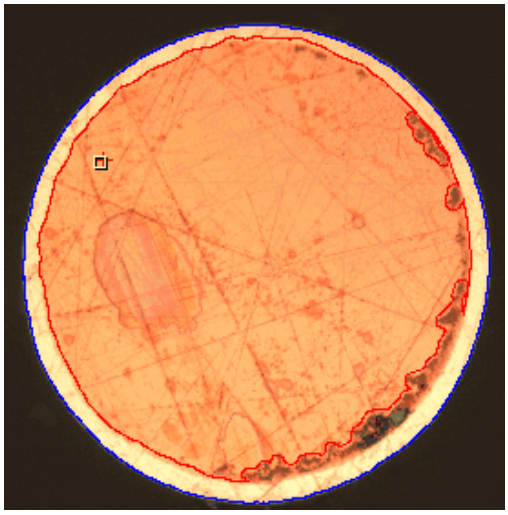
Edit borders step

1. Click the top *Add contours...* button.
2. Click the *Next* button.

Magic wand step

1. Then define the second contour. To do so, click the *Add contours...* button once more.
2. Then click a position inside the copper wire.
3. Take care again that the contour contains the inside of the copper wire as completely as possible, and that its outline isn't discontinued anywhere. At the same time, this new contour mustn't touch the contour that has already

been defined. Change the position of the slide control in the *Tolerance* field, until the second contour looks roughly as shown below:

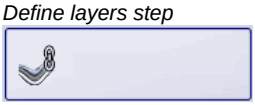


4. Click your right mouse button to finish the definition of the contour.
5. Click the *Next* button.

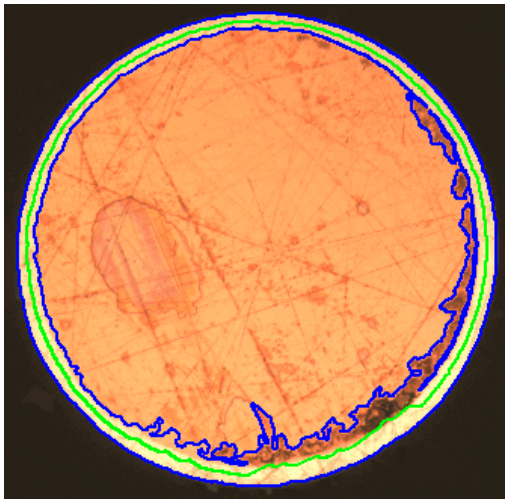
Edit borders step

1. Since you have already defined both of the borders, and don't want to change them: Click the *Next* button.

Define layers step



1. Click the *Add layers...* button.
2. Click the first border.
3. Click the second border.
   - The layer has now been defined. The neutral fiber is plotted in green. It always lies in the middle of the layer.



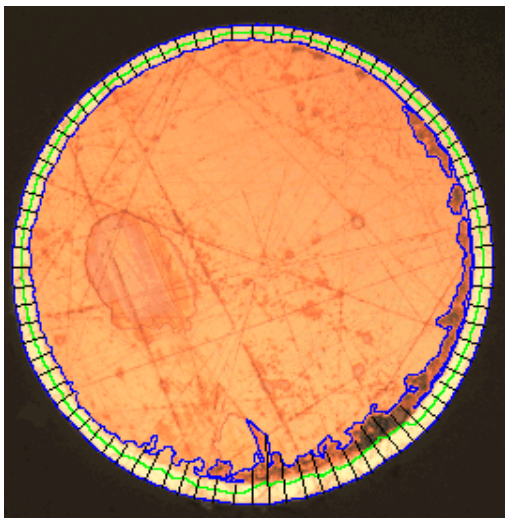
4. Click your right mouse button to finish the definition of the layer.
5. Click the *Next* button.

Image results step

1. Take a look at the results of the current image, shown in the *Image results* group.

- The values in the *Steps*, *Distance*, and *Type* fields can be edited when you double click in the cell you want to edit. You can find more information on this topic in the online help.
- The lower part of the group contains several buttons, with which you can change the way the layer thickness measurement is displayed. You can find more information on this topic in the online help.

2. Check the results shown in the image.
   - The measurement lines are shown in the image. To make them contrast better, the color of the measurement lines was set to black before the measurement took place.



3. Click the *Next* button.

Results step

1. Select the *Generate Report* check box, if you would like to have a report automatically generated in MS-Word once the analysis is completed.
   - The additional step *Reporting* will be added to the current analysis. In the lower part of the dialog box, the *Finish* button will change into the *Next* button.
2. Select the *Generate Workbook* check box, to have a document of the "workbook" type automatically created at the end of the analysis.
3. If you want to save the current settings to a file, click the *Save settings....* button. Then assign a descriptive name in the next dialog box.
   - You can load these settings (parameters) when you analyze further images. To do that for the new image in the *Image Source* step, click the *Load from file...* button.
4. Click the *Next* button.

Reporting step

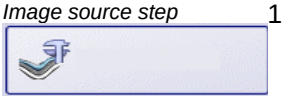
Define the report that contains the measurement results.

00730 12062012

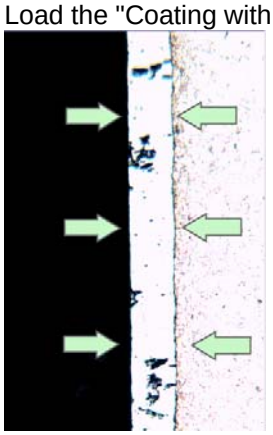
Performing a manual layer thickness measurement

Note: You can follow these step by step instructions on your PC. They describe a layer thickness measurement on an example image.

Image source step

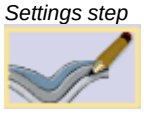


1. Load the "Coating with porosity.tif" example image.



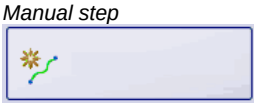
   - On this image, the middle layer is to be measured.
2. Activate the *Materials Solutions* tool window.
3. Click the *Layer Thickness* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
6. Select the *All images* entry in the *Check settings and results* list.
7. Click the *Next* button.

Settings step



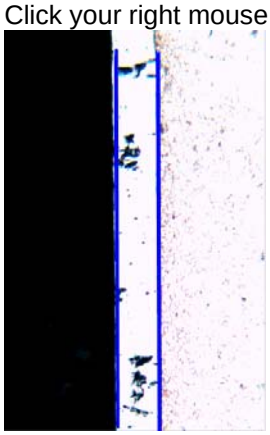
1. Click the *Manual* button.
2. In the *Layer type* group, click the icon for an open layer .
3. Click the *Next* button.

Manual step



1. Click the *Add borders...* button.
2. Define the first border. To do so, first click with your left mouse button at the position in the image where the border is to begin. Mark the course of the border with further left mouse clicks. Then click with your right mouse button at the position in the image where the border is to end.
   - The border will be shown in red.
3. Define the second border. To do this, proceed exactly as you did when you defined the first border.

4. Click your right mouse button to finish the definition of the two borders.



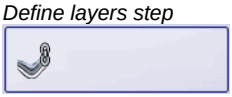
- The borders will be shown in blue.

5. Click the *Next* button.
   - The *Materials Solutions* tool window displays the *Edit borders* step.

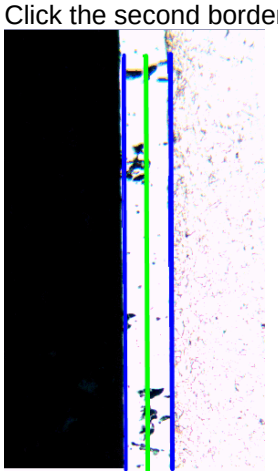
Edit borders step

1. Since you have already defined both of the borders, and don't want to change them: Click the *Next* button.

Define layers step



1. Click the *Add layers...* button.
2. Click the first border.
3. Click the second border.



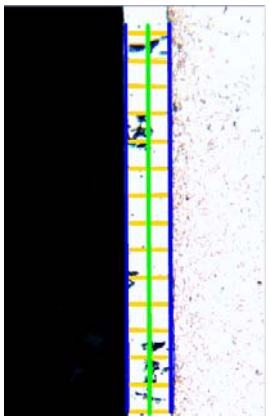
- The layer has now been defined. The neutral fiber is plotted in green. It always lies in the middle of the layer.

4. Click your right mouse button to finish the definition of the layer.
5. Click the *Next* button.

Image results step

1. Take a look at the results of the current image, shown in the *Image results* group. This group contains a table with the measurement results.
   - The values in the *Steps*, *Distance*, and *Type* fields can be edited when you double click in the cell you want to edit. You can find more information on this topic in the online help.
   - The lower part of the group contains several buttons, with which you can change the way the layer thickness measurement is displayed. You can find more information on this topic in the online help.

2. Check the results shown in the image.



   - The measurement lines are shown in yellow in the image.
3. Click the *Next* button.

Results step

Select the results you want.

Reporting step

Define the report that contains the measurement results.

00731 12062012

7.6. Cast Iron Analysis

What is a cast iron analysis?

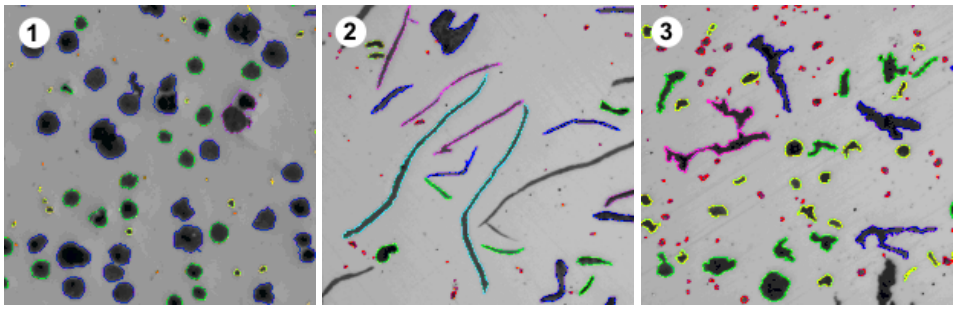
The quality and consistency of cast iron depends on the distribution and the morphology of its carbon content. By using a cast iron analysis you can determine the cast iron's graphite fraction with the help of unetched samples. As well as that, with the help of etched samples you can determine the ferrite/pearlite ratio.

The classification of the detected particles is performed according to the industrial standard that is selected in the program options. Each standard requires a different classification of the detected particles. These classifications are included in the software package purchased, and are automatically installed with it. The following standards are supported:

- ASTM A 247 2006
- EN ISO 945
- EN ISO 945-1
- KS D 4302-2006
- JIS G 5502-2001
- GBT 9441

Determination of the graphite fraction

By using your software's "Cast Iron" Solution, you can measure the graphite fraction and classify the detected particles. For this purpose, the sample must not be etched. How the classes are defined, depends on the standard according to which the cast iron analysis is carried out. The standard is set in the program options.



You see the results of a cast iron analysis made of different forms of graphite. The color coding of the particles indicates their belonging to a specific size class (**1**), form class (**2**), and a form factor (**3**).

Results of a cast iron analysis made to determine the graphite fraction

The results of an analysis can be displayed in a workbook. Additionally, or alternatively to that, the results can be displayed in an MS-Word report.

While you are performing a cast iron analysis, you can create a chart showing the graphite size, the graphite form or the graphite nodularity. You can also save these charts as files.

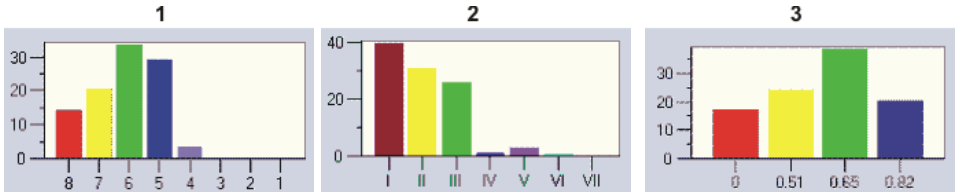


Figure (1) shows a chart of the graphite size. Along the X-axis the size classes are shown, along the Y-axis the relative number of detected particles in % is shown.
Figure (2) shows a chart of the graphite form. Along the X-axis the form classes are shown, along the Y-axis the relative number of detected particles in % is shown.
Figure (3) shows a chart of the graphite nodularity. Along the X-axis the form factor is shown, along the Y-axis the relative number of detected particles in % is shown.

General procedure for a cast iron analysis made to determine the graphite fraction

1. Select the analysis process

Click the *Cast Iron* button, located in the *Materials Solutions* tool window.

2. "Image source" step

Choose the image you want to measure.

3. "Settings" step

Choose the sample type (unetched).
Set the graphite parameters.

4. "Graphite Distribution" step

If chosen: Determine the distribution of the graphite particles.

5. "Image results" step

Take a look at your image results.
If necessary, Delete or separate detected particles, or add new ones.

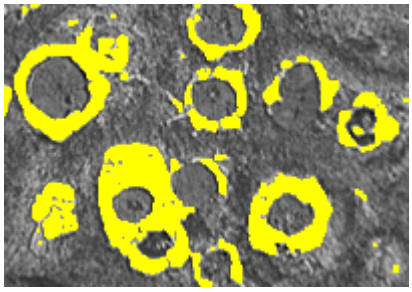
6. "Results" step

Document the results (report or workbook).

Determination of the ferrite/pearlite-ratio

By using your software's "Cast Iron" solution, you can also measure the ferrite/pearlite ratio. For this purpose, the sample must have been etched. Since graphite and pearlite have very similar gray values, it's difficult to differentiate between these two fractions in a sample during the same analysis. For this reason, determining the ferrite/pearlite ratio is done as follows:

To begin with, your software determines, by means of the definition of phases, the ratio of the bright ferrite areas to the dark (graphite and ferrite) areas. During the analysis, the graphite fraction is entered, and is then subtracted from the dark areas. This graphite fraction has either been determined in an earlier measurement (this value can then be imported), or it can alternatively be estimated. Using the pearlite area that has in this way been corrected, the ferrite/pearlite ratio is calculated.



You see a step in the analysis during the determining of the ferrite/pearlite ratio. The bright ferrite phase has been determined by your software (shown in yellow here).

00734 10052013

Performing a cast iron analysis (unetched sample)

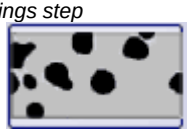
Note: You can follow these step by step instructions on your PC. They describe how the graphite fraction is determined.

Image source step

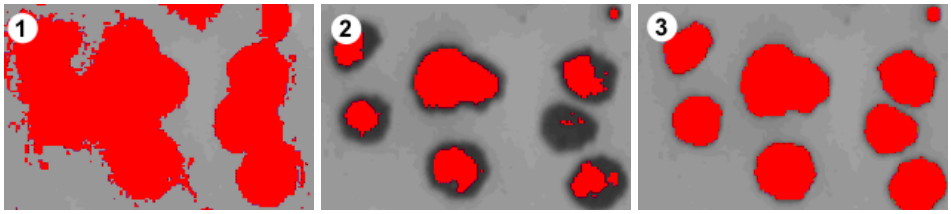
1. Load the "GlobularGraphite.tif" example image.
   - The graphite fraction is to be measured.
2. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
3. Click the *Cast Iron* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
   - By doing so, you skip the *Sample information* step which is not relevant for this example image. However, it is quite possible that, when performing your own analyses, you might want to load sample results (e.g., the result of a previous cast iron analysis that determined the graphite fraction). In this case, make sure the *Skip 'Sample information'* check box is cleared, which will enable you to use the *Load sample results* button in the *Sample information* step.
6. Select the *All images* entry in the *Check settings and results* list.
7. Click the *Next* button.

- The *Materials Solutions* tool window will display the next step.

Settings step



1. Click this button to set that you want to determine the graphite fraction in an unetched sample.
   - If the button for etched samples has been active before, the settings possibilities in this window will now change.
2. Use the slide control to define the threshold value for the graphite detection. Observe the sample. The threshold value has been correctly set when the graphite particles can be completely detected.



In the illustration (**1**), the threshold value has been set too high, the detected particles are too coarse. In the illustration (**2**), the threshold value has been set too low, the particles are not detected completely. The illustration (**3**) shows a correctly set threshold value.

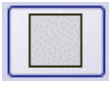
3. Select the graphite parameter that is to be determined. To do so, select the corresponding check box. The possibilities listed below are available: Which size classes, form classes and form factors are used for the classification, depends on the industry standard according to which the cast iron analysis is performed.
   - *Graphite size:* Sorts the detected particles into specific classes, according to their size.
   - *Graphite form:* Sorts the detected particles into specific classes, according to their form.
   - *Graphite nodularity:* Sorts the detected particles into specific classes, according to their nodularity. The nodularity is a unit of measure for the sphericity of the graphite.
   - *Graphite distribution:* Makes it possible to compare the distribution of the particles in the current image with the distribution in specific reference images. When this check box has been selected, the additional step, the *Graphite distribution* will be added to the cast iron analysis. The graphite distribution (types A-E) can only be determined for lamellar graphite.
4. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Graphite Distribution step

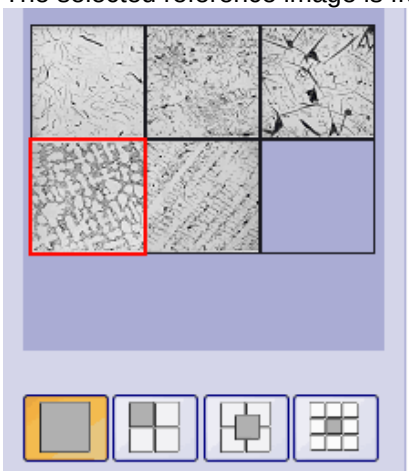
Prerequisite: You will only see this step if, in the previous step, you selected the *Graphite distribution* check box.

In this step, you can compare the particles that have been detected with reference images that show different distributions of graphite particles. You can then determine which of the reference images shows a distribution that is most similar to that of the current image. The reference images correspond to images that the chosen standard contains.

1. In the *Style* group, choose how the images are to be arranged in the document group for the comparison. Choose an arrangement in which the

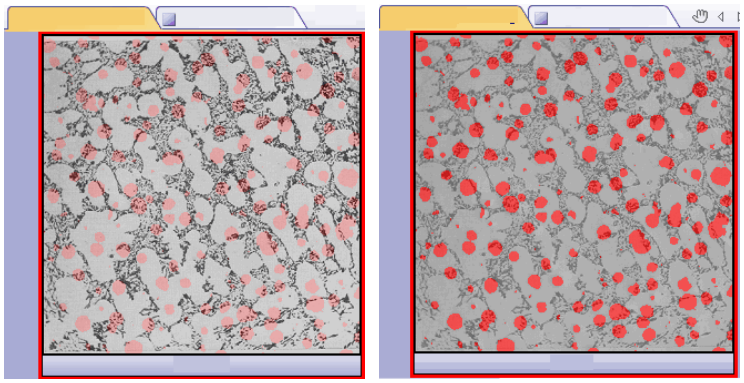
"GlobularGraphite.tif" image and the selected reference image are superimposed. To do this, click this  button.

- In the *Overview* field, you see the arrangement that has been chosen. The selected reference image is framed in red.



- The *Cast Iron Distribution* document will now be displayed in the document group. It contains exactly one image.

2. Compare the graphite distribution of the current image with that of the reference image. Move the slide control below the *Style* field towards the *Opaque* position, if the image that is to be checked is to superimpose the reference image. Alternatively, move the slide control towards the *Transparent* position, if the image that is to be checked is to be superimposed by the reference image. If you want to choose another reference image, in the *Overview* field, click that image with your left mouse button.



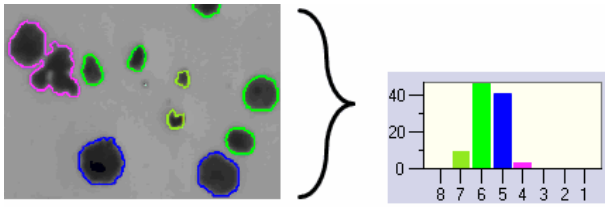
The illustration on the left shows the image that is to be checked. Because the slide control is located in near the *Opaque* position, the reference image's structures can only be faintly recognized. For the illustration on the right, the slide control has been moved towards the *Transparent* position. Now, the reference image can be clearly recognized, and the image that is to be checked can be only faintly recognized.

3. When you have selected the reference image that is the most similar to the image that is to be checked: Click the *Accept* button.
   - The chosen image's data will be accepted in the *Results* field.
   - It's possible to accept several reference images, for example, with samples that have very different structures.
4. Click the *Next* button.

- The *Materials Solutions* tool window will display the next step.

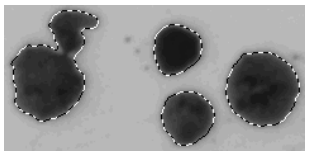
Image results step

1. Take a look at the results that are shown in the table and also in the image. Select the *Show graphite detection* check box, in the *Validation* group.
   - Every particle that has been detected will then be outlined with a colored line. The color with which the particle is outlined, shows you to which class it belongs. The same colors will be used in the chart.



On the left, you see the colored identification of the particles in the image. On the right, you see the chart of graphite sizes, that uses the same colors.

   - Particles that have been detected, but that aren't used for the analysis (e.g., because they don't come up to the minimum size that has been set for the program options), are shown with a dashed line.



2. If you selected several graphite parameters in the *Settings* step, toggle between the different charts.
3. If you want to correct the automatically found particles, use the buttons in the *Validation* group. You will find step-by-step instructions on how to correct particles here.
4. Click the *Next* button.

Results step

1. Take a look at the results that are shown in the table. Among other things, the number of particles is shown here.
2. Select the *Generate Report* check box, if you would like to have a report automatically generated in MS-Word once the analysis is completed.
3. Select the *Generate Workbook* check box, to have a document of the "workbook" type automatically created at the end of the analysis.
   - Leave the *Create chart* check box cleared for these step-by-step instructions.
4. Click the *Save sample results* button, if you want to also determine the ferrite/pearlite-ratio in another cast iron analysis, on the basis of the etched sample. You can then load the graphite fraction determined here, and won't need to enter it manually.
5. Click the *Next* button.

Reporting step

Define the report that contains the measurement results.

- The *Materials Solutions* tool window switches back to the start position. You can now use all of your software's functions again.
- Through the materials analysis measurement, the image has collected one or more additional layers (can be seen in the *Layers* tool window). If

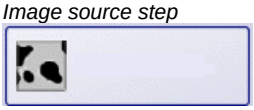
required, save the image in TIF or VSI format to retain these newly created image layers.

00736 12062012

Performing a cast iron analysis (etched sample)

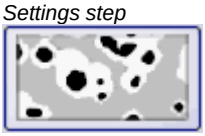
Note: You can follow these step by step instructions on your PC. They describe how to measure the ferrite/pearlite-ratio.

Image source step

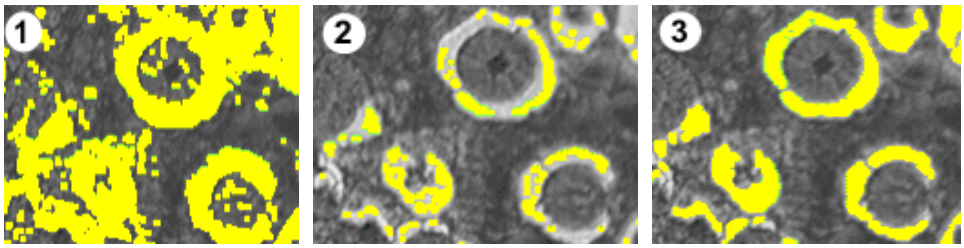


1. Load the "Ferrite Pearlite.tif" example image.
   - The ferrite/pearlite-ratio is to be measured.
2. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
3. Click the *Cast Iron* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
6. Select the *All images* entry in the *Check settings and results* list.
7. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Settings step

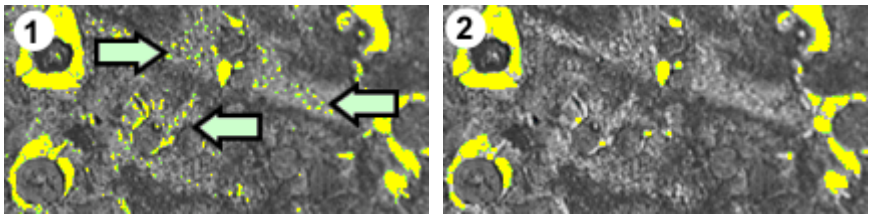


1. Click this button to set that you want to determine the ferrite/pearlite-ratio, using an etched sample.
   - If the button for unetched samples has been active before, the settings possibilities in this window will now change.
2. Use the *Threshold for ferrite* slider to define the ferrite phase. By doing so, you set the range of intensity values (the phase) that is valid for the ferrite detection. If the slide control is closer to the *Low* position, the phase contains a larger part of the intensities that are present in the image. If the slide control is closer to the *High* position, the phase contains a smaller part of the intensities. This means that only a smaller part of the intensity values is detected as ferrite. All of the pixels that have been detected as ferrite will be highlighted in yellow in the image.
   - The threshold value has been correctly set when the ferrite is completely detected.



In the illustration (**1**), the threshold value has been set too high, too many particles are detected as ferrite. In the illustration (**2**), the threshold value has been set too low, the ferrite is not detected completely. The illustration (**3**) shows a correctly set threshold value.

3. Use the *Closing pearlite phase* slide control, to define how rigid the "voids" that the pearlite contains, are to be closed. In this context, a void in the pearlite is an area within the pearlite that has so bright intensity values, that it is assigned to the ferrite. In the image, voids are visualized as an accumulation of small yellow points within the pearlite.
   Using the *Closing pearlite phase* slide control is a means of correcting these voids. To do so, a morphological filter is applied. Morphological filters are often used in image analysis to optimize the results of an automatic object analysis.

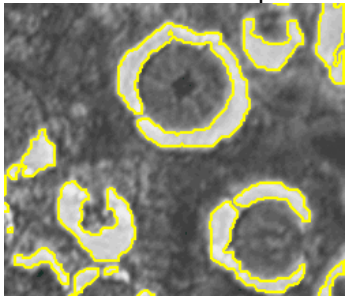


In the illustration (**1**), the pearlite phase is little closed. This is why many voids have been detected within the pearlite (see arrows). The illustration (**2**) shows a pearlite phase that is more closed.

4. In the *Graphite fraction* group, select how this sample's graphite fraction is to be entered. The graphite fraction will be subtracted from the detected pearlite fraction. Using the pearlite area that has in this way been corrected, the ferrite/pearlite ratio is calculated. This step is necessary because graphite and pearlite have very similar gray values and can therefore not be detected separately by the software. There are two possibilities how to enter the graphite fraction:
   - You select the *Enter manually* option and enter the value. This option is always active. You can have e.g., made a note of this value, or have saved it in a report.
   - You select the *Result of unetched sample analysis* option. This option is only active if, in the same analysis, you have already measured the graphite fraction, using an unetched part of the sample. This option is also active if, you measured the graphite fraction in a previous analysis, saved these values in a parameter set and loaded them in the current analysis' *Sample information* step.
5. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Image results step

1. Take a look at the results that are shown in the table. Among other things, here you will find the ferrite/pearlite-ratio that has been measured.
2. Take a look at the displayed results in the image as well. To do so, select the *Show ferrite detection* check box, in the *Validation* group.
   - Each detected ferrite particle will now be outlined in yellow.



3. Click the *Next* button.

Results step Select the results you want.

Reporting step Define the report that contains the measurement results.

00737

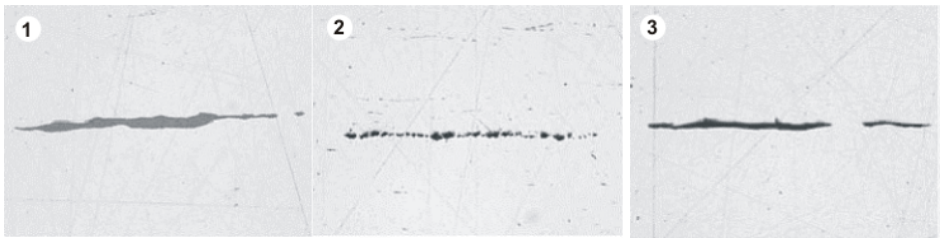
7.7. Inclusions Worst Field

What is an inclusion worst field analysis?

An inclusion worst field analysis is one of several possible procedures used to detect non-metallic inclusions in metal samples. This analysis is, e.g., used to measure the amount, size and distribution of sulfides and oxides in steel. With the measurement results, different production processes can be compared, or the quality of a product determined.

What exactly is a non-metallic inclusion?

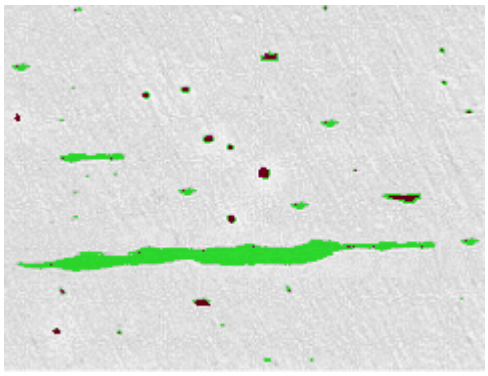
During the production processes, non-metallic inclusions can accrue within steel alloys. Inclusions affect the chemical and mechanical properties of the steel. The fewer inclusions there are in a steel, and the smaller and homogeneous these are, the better is its quality.



Microscope image of different inclusions in a polished steel sample. The inclusions differ in their color and form. The images show a sulfide inclusion (**1**), a silicate inclusion (**2**), and an aluminum inclusion (**3**).

The nature and appearance of the non-metallic inclusions depend on a variety of factors, such as, e.g., the steel type, or the production process. The inclusions are divided into different classes according to their appearance (color, form, and size). The classification is made according to different industry standards.

Since all inclusions are darker than the color of the steel, they can easily be detected by means of an automatic image analysis. When detecting the inclusions, the inclusion worst field analysis searches for particles. For the image analysis software, a particle is a cohesive number of pixels, that all lie within a defined intensity range. For this reason, you first have to define the intensity range. Since between the different inclusions there are also intensity differences, (sulfides are, e.g., brighter than oxides), you can also define two intensity ranges.



Particle detection during an inclusion worst field analysis. When a suitable definition of the gray value ranges has been made, the sulfides (green) and the oxides (red) will be detected.

Editing inclusions

You can manually edit the inclusions that your software found automatically. You have the possibility of deleting, splitting, or joining up inclusions, and you can also change their type.

Results of an inclusion worst field analysis

The inclusion worst field analysis determines which is the largest non-metallic inclusion within the sample under investigation. This is done for each inclusion type separately. The classification and naming convention of the inclusions differs from industry standard to industry standard. The sizes are measured in accordance with the industry standards:

- ASTM E 45 Method A
- DIN 50602 Method M
- ISO 4967 Method A
- GB/T 10561 Method A
- JIS G 0555 Method A
- UNI 3244 Method M
- EN 10247 Method M(L/n)
- EN 10247 Method M(L/d)
- EN 10247 Method M(L/d)
- EN 10247 Method M(A/n)
- EN 10247 Method P(A)
- EN 10247 Method P(L/d)

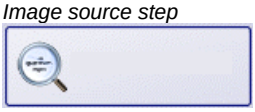
The results of an analysis can be displayed in a workbook. Additionally, or alternatively to that, the results can be displayed in an MS-Word report.

00733

Performing an inclusions worst field analysis

Note: You can follow these step by step instructions on your PC. It describes how you can detect the worst inclusion in a sample.

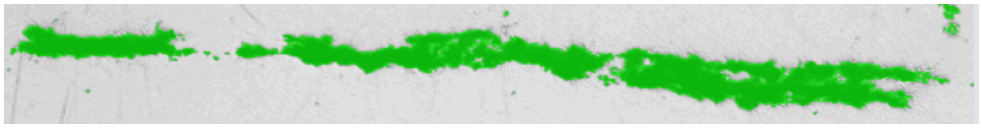
Image source step



1. Load the NMI0_0.tif example image.
   - The largest non-metallic inclusion is to be measured.
2. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
3. Click the *Inclusions Worst Field* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
6. Select the *All images* entry in the *Check settings and results* list.
7. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Settings step

1. In the *Evaluation method* field, set the standard you are going to use for the analysis.
2. Use the slide control to define the threshold value for all of the inclusions. Observe the sample. The threshold value has been correctly set when the inclusions are completely recognized.

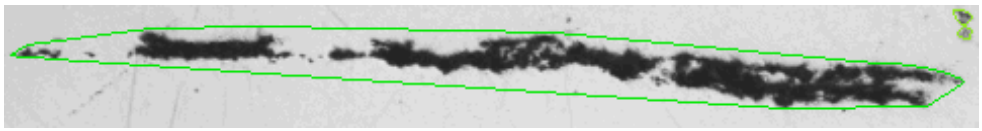


The illustration shows a correctly set threshold value.

3. Since in this sample there are no oxide inclusions, set the *Threshold oxide inclusions* slide control at the position *Low*.
4. Select the *Show ignored particles* check box, when you want to also have the particles that weren't included in the analysis, shown.
   - Particles will be ignored when they don't fulfill the requirements that were specified in the program options.
5. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Image results step

1. Take a look at the results that are shown in the table. Should you have analyzed several images of the same sample, you can switch between a display of the image results for the current image and the results for all of the images. To do this, select either the *Image* option, or the *Sample* option, located below the table.
   - The table with the measurement results contains a classification of the inclusions that have been detected. How this classification looks like, depends on the standard by which the analysis was performed. For example, the "ASTM E 45 Method A" standard uses the classification A (Sulfide), B (Alumina), C (Silicate) and D (Globular Oxide). Furthermore, this standard groups the inclusions into "t" (thin) and "h" (heavy), according to their mean width (inclusions type A, B, C) or to their diameter (inclusions type D). Other standards use another classification of the inclusions, and don't further divide them up into groups.
2. Take a look at the displayed results in the image as well.
   - In the image, every detected inclusion will now be outlined with a colored line.



The illustration shows a detected particle. The entire inclusion is outlined with a colored line.

   - Particles that have been detected, but that aren't used for the analysis (e.g., because they don't come up to the minimum size that has been set in the industry standard), are shown with a yellow line.
3. If you want to correct the automatically found inclusions, use the buttons in the *Edit inclusions* group.
4. Click the *Next* button.

Results step

1. Take a look at the results that are shown in the table. Here, you see, for each inclusion type separately, the worst inclusion found in any of the analyzed images.
2. Select the *Generate Report* check box, if you would like to have a report automatically generated in MS-Word once the analysis is completed.
3. Select the *Generate Workbook* check box, to have a document of the "workbook" type automatically created at the end of the analysis.
4. If you want to save the current settings to a file, click the *Save settings....* button. Then assign a descriptive name in the next dialog box.

Reporting step

Define the report that contains the measurement results.

00738 28062012

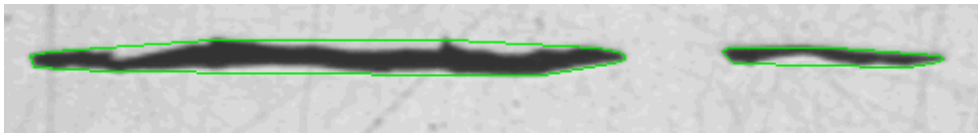
Editing inclusions

You can manually edit the inclusions that your software found automatically. In the *Image results* step, you have the possibility of deleting, splitting, or joining inclusions, and you can also change their type.

Please note: If you have manually corrected inclusions and return to the *Settings* step (e.g., to change the settings of the slide controls), your manual corrections will be deleted.

Merging inclusions

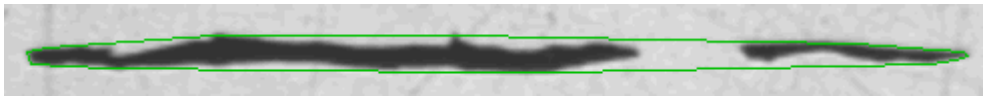
1. Enlarge the display of the image until you can easily recognize the two inclusions that you want to join.
   - In this example, these two inclusions are to be joined.



2. In the *Edit inclusions* group, click the *Merge inclusions* button.
   - The mouse pointer will change its form. You will then be in edit mode. The only thing you can do now is to add inclusions. In this mode, other work with your software isn't possible.
3. With your left mouse button, click the two inclusions.

Should you join two inclusions that belong to different inclusion types, the inclusion type of the first inclusion you selected will be used for the new joint inclusion. In this case, take care that you click the two inclusions in the right order.

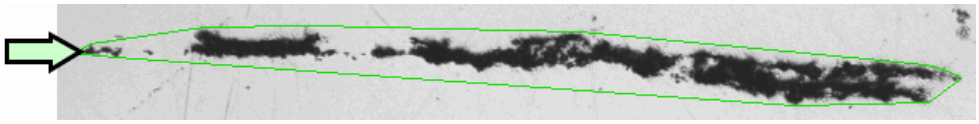
   - The inclusions will be joined. The results will be updated.



4. If you want to, you can merge further inclusions.
5. Click your right mouse button to leave the edit mode, and to accept the changes.

Splitting an inclusion

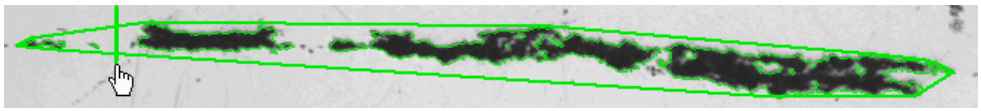
1. Enlarge the display of the image until you can easily recognize the inclusions that are to be separated.
   - In this example, the particle on the far left-hand side that is indicated by an arrow is to be split.



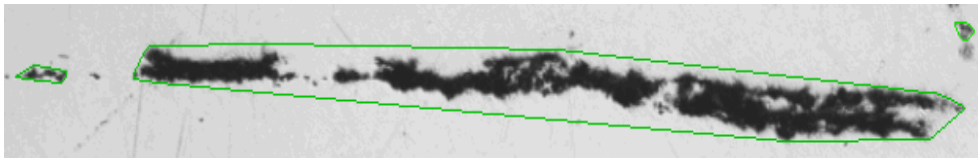
2. In the *Edit inclusions* group, click the *Split inclusion* button.
   - The mouse pointer will change its form. You will then be in edit mode. The only thing you can do now is to split inclusions. In this mode, other work with your software isn't possible.
3. To do so, click once with your left mouse button at an arbitrary position on the line surrounding the inclusion.

- The surrounding line and all of the particles that belong to this inclusion will be displayed bold.

4. Click with your left mouse button at the position in the image where the separation line is to begin.
5. While keeping your left mouse button depressed, drag the separation line through the object.



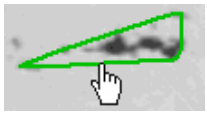
6. Click the left mouse button to confirm the separation.
   - The inclusion will be split.



7. If you want to, you can split further inclusions.
8. Click your right mouse button to leave the edit mode, and to accept the changes.
   - The results will be updated.

Deleting an inclusion

1. Enlarge the display of the image until you can easily recognize the inclusion that is to be deleted.
2. In the *Edit inclusions* group, click the *Delete inclusion* button.
   - The mouse pointer will change its form. You will then be in edit mode. The only thing you can do now is to delete inclusions. In this mode, other work with your software isn't possible.
3. Position your mouse pointer on the inclusion that is to be deleted.
   - The line surrounding the inclusion will be displayed bold.



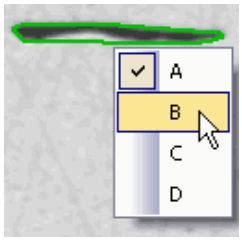
4. Click the left mouse button.
   - The inclusion will be deleted.

5. If you want to, you can delete further inclusions.
6. Click your right mouse button to leave the edit mode, and to accept the changes.
   - The results will be updated.

Changing the inclusion type

1. Enlarge the display of the image until you can easily recognize the inclusion that is to be changed.

2. In the *Edit inclusions* group, click the *Change inclusion type* button.
   - The mouse pointer will change its form. You will then be in edit mode. The only thing you can do now, is to change an inclusion's type. In this mode, other work with your software isn't possible.
3. Position your mouse pointer on the inclusion whose type is to be changed.
   - A picklist will drop down. It shows all of the inclusion types that the currently chosen standard contains. A check marks the currently chosen inclusion type.



An example of what the picklist can look like. Depending on the standard that has been chosen, the picklist can contain other entries.

4. Select the new type of inclusion you want.
   - The new inclusion type will be assigned. In the image, the inclusion will now be displayed with a surrounding line in another color.

5. If you want to, you can change the type of further inclusions.
6. Click your right mouse button to leave the edit mode, and to accept the changes.
   - The results will be updated.

00739

7.8. Throwing Power

What is a throwing power measurement?

Use the *Throwing Power* solution to determine the quality of the copper plating on an HDI panel. You can measure through-holes, microvias and filled microvias.

The *Throwing Power* solution is completely integrated into the *Materials Solutions* tool window. The tool window works in a similar way to a software wizard. As soon as you've started an analysis process you'll be guided step by step through the measurement.

Before you start a throwing power measurement

The following conditions must be met before you start a throwing power measurement.

1. Prepare suitable cross-sections through the panels. To measure a through-hole, you will also need a flat section of it.
2. The results of a throwing power measurement will always be saved in a database. You should thus open the required database.
   If no database exists yet, create one using the database template supplied. You can find more information in the online help.
3. Align your microscope.
4. Make sure that your software is correctly configured. You can find more information in the online help.
5. Start your software. Switch to live mode and select the best settings to acquire an image. While a throwing power measurement is in progress, you may no longer alter all settings for the image acquisition.
   - Check the white balance If necessary, carry out a white balance. You can find more information in the online help.
   - Select the live-image's resolution in the *Camera Control* tool window.

General procedure for a throwing power measurement

1. Select the analysis process

Click the *Throwing Power* button, located in the *Materials Solutions* tool window.

▼

2. "Settings" step

Select one of the following measurement methods: *Through-holes*,*Microvias*, *Filled through-holes*, *Filled microvias*.

▼

3. "Diameter" step

If you have selected the *Through-holes* measurement method: Place a flat section of the through-hole under your microscope. Measure the diameter of the through-hole.

▼

4. "Measurements" step

Place a cross section through the panel under your microscope. Measure all of the parameters specified in the *Measurements* list.
Repeat this step in the analysis and measure other cross sections.

▼

5. "Report images" step

Acquire three images for the final report.

▼

6. "Reporting" step

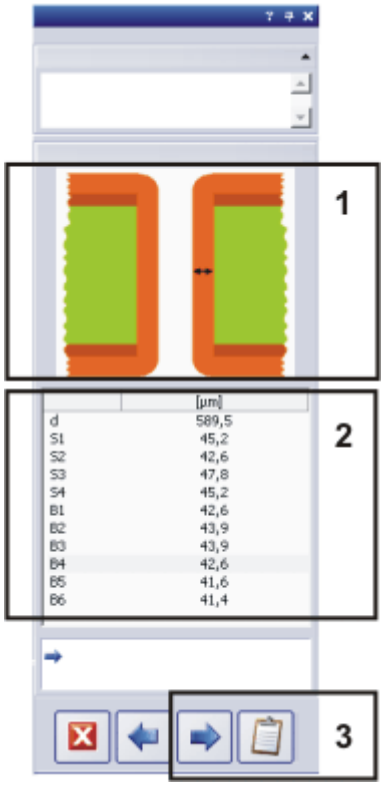
Produce a report with the results of the throwing power measurement.

00500 25062012

Defining throwing power measurements

In this step you carry out the actual throwing power measurement. The sequence and the measurement parameters are specified. Your software will show all of the required measurement parameters in a diagram.

The following options are available:

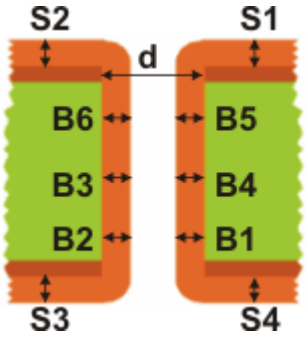
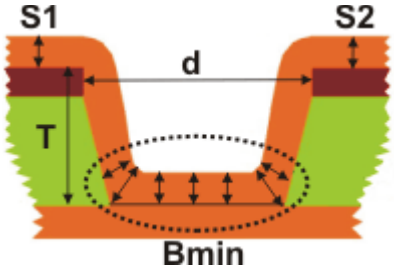
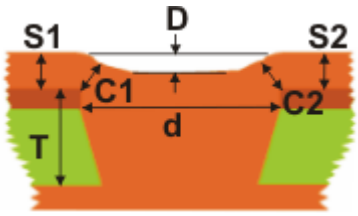
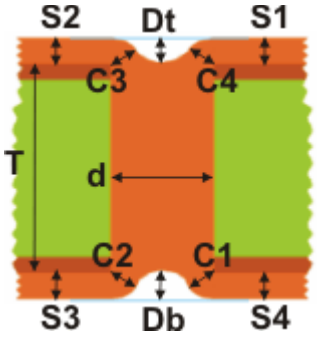


(1) Displaying the parameters to be measured
(2) Performing the throwing power measurement
(3) Continuing the throwing power measurement

(1) Displaying the parameters to be measured

Your software will provide you with visual support when performing the throwing power measurement. The distance which is to be measured next will be drawn in the schematic illustration at the top of the *Materials Solutions* tool window. The copper plating is shown in orange, the panel in green.

From the *Measurements* list, choose any measurement parameter to show the distance to be measured in the schematic illustration.

Measurement method	Schematic illustration	Required measurement parameters
Through-holes		**d** = diameter of the through-hole (additional remarks further below) **S1-4** = thickness of the surface plating **B1-6** = thickness of the plating inside the through-hole
Microvias		**d** = diameter of the microvia **T**= thickness of the panel **S1**, **S2** = thickness of the surface plating **Bmin** = minimum thickness at the bottom of the microvia (additional remarks further below)
Filled microvias		**d** = diameter of the microvia **T**= thickness of the panel **S1**, **S2** = thickness of the surface plating **C1**, **C2** = minimum corner plating thickness **D** = the dimple's height or depth (additional remarks further below)
Filled through-holes		**d** = diameter of the microvia **T**= thickness of the panel **S1-4** = thickness of the surface plating **C1-4** = minimum corner plating thickness at the top and bottom of the microvia **Dt** =the dimple's height or depth at the upper side **Db** = the dimple's height or depth at the lower side (additional remarks further below)

Remarks about the required measurement parameters

1. You can't precisely measure the actual **through-hole diameter** on the panel's cross section if the cross section doesn't run exactly through the center of the hole. Therefore, a separate measurement of the hole's diameter is needed on a flat section of the through hole.
2. Look for the minimum plating thickness in the area that is indicated in the illustration. Measure this thickness to get the**Bmin** parameter.
3. A **Dimple** represents the difference in heights between the plated copper within a microvia or through-hole and around the perimeter of that microvia or through-hole. When the filled microvia or through-hole is not completely filled, the measurement value is positive.

When the filled microvia or through-hole is overfilled, the measurement value is negative.

(2) Performing the throwing power measurement

Measure the distance which is shown in the diagram. To do that, click with the left mouse button on the start and end points of the distance to be measured.

As soon as you have measured all of the parameters which are required for the selected measurement method, the *Next* and *Get Results* button will become active in the lower part of the dialog box.

(3) Continuing the throwing power measurement

As soon as you have measured all of the parameters which are required for the selected measurement method, the *Next* and *Get Results* button will become active in the lower part of the dialog box.

Performing additional measurements

Click on the *Next* button to measure more structures on your panel. All of the measurement parameters will be reset for the new measurement. The previously measured parameters will be saved and will be included in the analysis, which will be issued in a report at the end of the throwing power measurement. If you would like to examine the individual measurements again later on, save a workbook in the database in the *Reporting* step.

Finishing the throwing power measurement

Click the *Get Results* button. With this, the actual throwing power measurement is finished. Acquire pictures for the report now and then create the report.

Possible warnings once the measurement is complete

To get statistically reliable measurement results, you will have to measure several though-holes or microvia. You can set a required minimum number of between 1 and 10. To do this, use the *Tools > Options > Materials Solutions > Throwing Power* dialog box. Three measurements is preset as the minimum number.
If you finish the measurement before that, a corresponding warning message will be produced.

During the analysis, the arithmetic mean of all of the measurements carried out will be calculated. If the standard deviation for a measurement parameter exceeds 5%, you will also be presented with a corresponding warning. In this case, measure another five structures on the panel, to increase the statistical reliability.

10594 25062012

Performing a throwing power measurement

Example: these step-by-step instructions describe the *Microvias* measurement method as an example for a throwing power measurement. The other measurement methods which are available work in a similar way.

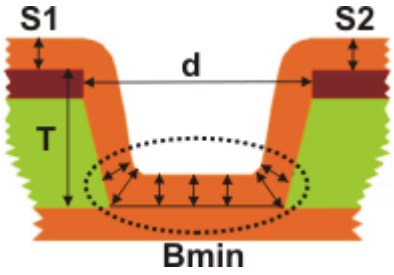
1. Prepare the throwing power measurement. Note all of the requirements for this, listed here.
2. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.

3. Click the *Throwing Power* button.
   - The *Materials Solutions* tool window displays the *Settings* step.

Selecting the measurement method

4. From the *Measurement method* list, select the*Microvias* entry.
   - In the *Materials Solutions* tool window you will now see a schematic illustration, which shows the cross section through a filled microvia.

5. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.
   - Your software will automatically switch to the live mode.
   - The *Camera Control* and *Microscope Control* toolbars will be displayed so that you can set the exposure time and the current magnification.
   - The *Measurements* table in the *Materials Solutions* tool window contains the required parameters to measure the throwing power of a microvia. The first measurement parameter **d** will automatically be selected and will be shown in the schematic illustration in a tool window.



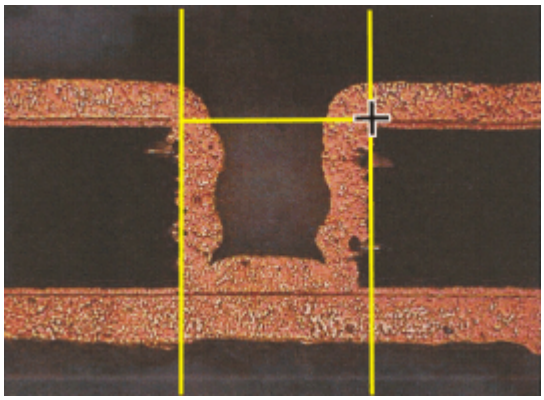
The *Microvias* measurement method contains the following parameters: **d** = diameter of the microvia, **T** = thickness of the circuit board, **S1** and **S2** = thickness of the surface coating of the track, **Bmin** = minimum thickness of the coating at the base of the microvia

Measuring the first microvia

6. Place one of the circuit board cross sections to be measured under your microscope. Refer to the cross section as shown in the schematic illustration.
7. Move the stage so that the first distance to be measured can be clearly seen in the live image and adjust the focus.
8. Select the best magnification. To do that, in the *Microscope Control* toolbar, click the required objective's button.

Note: The images which you acquire with your software will only be correctly calibrated if you specify the current objective magnification before acquiring the image. Correctly calibrating the image is a requirement for a correct measurement.

9. Set a zoom factor for your image window that will make the through-hole that is to be measured clearly visible. Rotate, for example, the mouse wheel to change the zoom factor of the live image in the image window.
   You will achieve the most precise measurements if you set the zoom factor at 100%.
10. If necessary, adjust the exposure time.
11. Measure the distance which is shown in the diagram. To do that, click with the left mouse button on the start and end points of the distance to be measured.
    - The line you have measured will be displayed in the image.
    - The result will be displayed in the *Materials Solutions* tool window in the *Measurements* table.
    - Your software will now automatically activate the next parameter to be measured in the *Measurements* table and will also show this in the diagram.



In the live image, measure the diameter of the microvia. The two helper lines stand vertically on the measured distance and help you to align the measured distance exactly with the microvia's borders.

12. Move the stage so that the next distance to be measured can be clearly seen in the live image and adjust the focus.
    If necessary, select another objective magnification to be able to measure the distance with the best accuracy.
13. Measure the required distance.
14. Repeat the last steps until you have measured all of the required parameters. For the last measurement parameter, Bmin, measure the lowest thickness of the coating within a certain area. This area is circled in the schematic illustration.
    - As soon as you have measured all of the parameters which are required for the selected measurement method, the *Next* and *Get Results* button will become active in the lower part of the dialog box.

Measuring the other microvias

15. Click the *Next* button to conclude the measurement for the current microvia.
    - The *Materials Solutions* tool window displays the *Measurements* step.
    - As an intermediate step, your software will save all of the values measured so far.
    - All values from the last measurement will be deleted from the *Measurements* table.
16. Now measure the next microvia. To get statistically reliable measurement results, you will have to measure several microvia.

Finishing the throwing power measurement

17. Click the *Get Results* button, when you have measured the required number of microvias.
    - The *Materials Solutions* tool window displays the *Report images* step.
18. Acquire three images to document the measurement. You could for example, acquire three different cross sections at a low magnification. Or you acquire an overview image of a microvia and then acquire two images at a higher magnification showing interesting details.
    If necessary, change the sample for this. Move the stage to the required location. Select a suitable magnification and exposure time and focus on the sample.

    Click on this button , to acquire the image.
    - The images acquired will be displayed in the *Materials Solutions* tool window.

Acquire three images to finish the throwing power measurement.

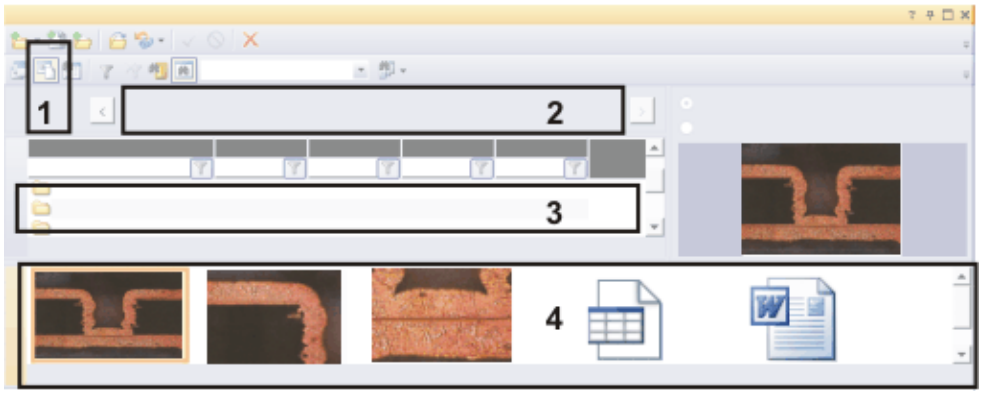
19. Open the database where you would like to save the measurement results. In the database, select the folder where the measurement results are to be saved, or create a new record.
Additional information on inserting data into a database can be found in the online help.

20. Click the *Next* button.
    - The *Materials Solutions* tool window displays the *Reporting* step. In the *Template* group, you'll see a preview of the document template that has currently been chosen.
21. Select the *Add workbook to the database* check box.
22. Start the MS Word application.

23. Click this button.
    - With this, the throwing power measurement is finished.
    - The *Materials Solutions* tool window switches back to the start position.
    - The three images acquired will be saved in the database. The name of the images in the database is determined by the current default value for the *Image name* database field. The database administrator can set up this default value. You can find more information in the online help.
    - A workbook with the measurement results will be created and saved in the database.
    - The concluding report will be generated and displayed in MS-Word.

Editing the report and saving it

1. Check the report in MS Word. If necessary, add more text.
2. When you are satisfied with the report, in MS Word, use the *Olympus > Save to Database* command to also insert the report into the database. Before doing that, make sure that the correct database folder has been selected.



The results of a throwing power measurement will be saved in the database. You can for example access the data in the document view of the database (**1**). The project header view (**2**) shows the higher level database folder. In the sample list view (**3**) the database folder will be selected which contains the data. The gallery view (**4**) shows the three images acquired, the workbook with the measurement results and the saved MS Word report.
You can find additional information on the document view of your database in the online help.

Loading the measurement results

3. In your software, open the database where the measurement results are saved.
4. Switch to your database's document view. Select the database folder which contains the measurement results. If your database is based on the supplied database template, then the database folder is a record of type *Sample*.
5. In the gallery view, double click, for instance, on the symbol for the workbook to view the measurement results.
   - The workbook contains the values for all of the microvias measured.
   - Statistical values e.g. standard deviation, can also be displayed in the workbook. You can specify exactly which statistical values will be shown. To do this, open the *Tools > Options > Measurement and ROI > Results* dialog box.

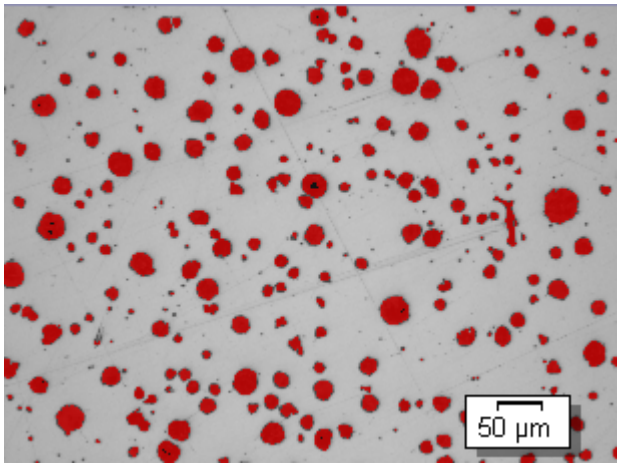
00501 25062012

7.9. Porosity

What is a porosity measurement?

With a porosity measurement, you measure the percentage of the surface which is made up by pores in your samples as well as the number and density of the pores.

It is a precondition for the measurement that the pores differ from the rest of the sample, e.g. because they are darker or lighter. The pores thus have differing intensity values to the rest of the sample, making automatic analysis of the image possible. For the image analysis, so-called phases are defined which cover a certain range of intensity values. If the pores largely have the same intensity values, one phase will be sufficient. If the pores have very different intensity values, then several phases will have to be set up.



Porosity measurement on one image. All of the pixels which lie within the defined intensity range, will be shown in color during this step in the analysis. In the example shown, red has been selected for the phase.

The result of the automatic image analysis can be restricted by the definition of counting conditions. Pores which do not meet the counting conditions will not be considered when determining the pore density and their number.

Manually processing the image porosity value

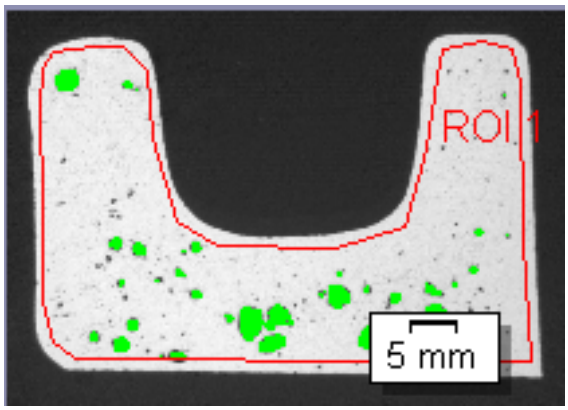
You can manually adjust the result of the automatic image analysis. You do this interactively on the image. Note that you are not changing the image itself, but the image's measurement layer.

You can manually delete parts of the image which have been recognized as pores (in image analysis, one speaks of "detected objects"). This can be necessary, if for example artifacts in the image are recognized as pores because they have similar intensity values. By manually deleting these objects, the artifacts will be excluded from the analysis.

In addition, you can also manually add other image segments which were not detected as such but which are actually pores. With the manual addition and deletion of objects, you always change the percentage porosity value of the image.

Measuring on ROIs

You can choose whether you would like to measure the entire image or if the measurement should only be carried out on a part of the image, a so called ROI (Region Of Interest). You can also define several ROIs.



On the image, the porosity of an ROI is measured.

Results of a porosity measurement

The results of an analysis can be displayed in a workbook. Additionally, or alternatively to that, the results can be displayed in an MS-Word report.

General procedure for a porosity measurement

1. Select the analysis process

Click on the *Porosity* button, located in the *Materials Solutions* tool window.

▼

2. "Image source" step

Choose the image you want to measure.

▼

3. "ROIs" step

Define the ROIs, or measure the porosity of the whole sample.

▼

4. "Threshold" step

Define the intensity range for the first phase, and define other phases if necessary.

▼

5. "Counting conditions" step

Define the pore size parameter and the minimum and maximum pore size.

▼

6. "Image results" step

Take a look at your image results.
If necessary, delete pores, or add new ones.

▼

7. "Results" step

Document the results (report or workbook).

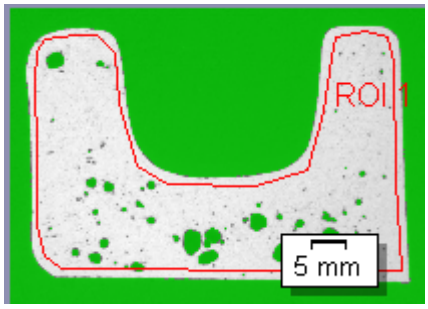
10606 04062012

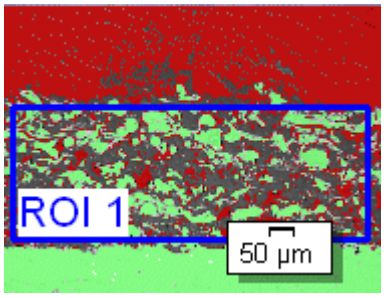
Thresholds for porosity measurements

All of the pixels which lie within an automatically defined intensity range, will be shown in color during this step in the analysis. This intensity range is called a "phase".

The intensity value range is limited by a top and a bottom intensity value. These are the so-called threshold values.

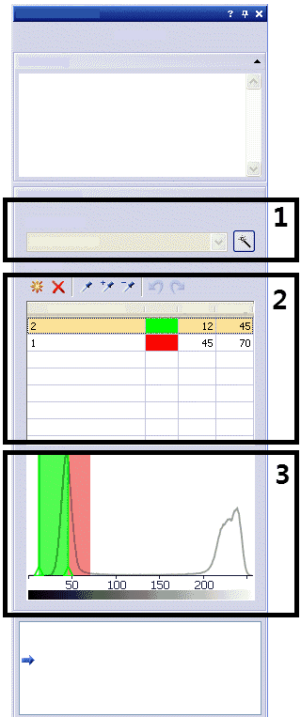
In this step in the analysis you can change the threshold values. You can also create another phase. This makes sense, for example, if the pores have very variable intensity values.





To the left you can see a sample which only has one phase. On the right you can see a sample with a red and a green phase.

Please note that defined ROIs will not be considered in this step in the analysis, but only in the next step.



(1) "Component" field

Click on the *Automatic Threshold Computation* button, to have the threshold values initially calculated automatically, to then manually process them, if necessary. The *Compute automatic thresholds* dialog box will open. Make the necessary settings here.

If you measure the porosity in a color image, in the *Component* list you can select whether the threshold value is to be determined on the intensity value or on the red, green or blue part of the image. The threshold value setting in color images is more complex than it is in gray-value images. You will find detailed step-by-step instructions on setting threshold values on color images in the online help.

(2) Defining threshold values

Note: If you would like to set threshold values for several phases in a gray-value image, you will have to begin by setting the threshold value for the darkest phase. Then set the threshold value for the next brightest phase, and so on.

Click on the *New Threshold* button, to set an initial value for the selected phase's threshold value range. As soon as you move your mouse pointer onto the image it will change its shape to that of a pipette.
Click on one pixel or on the image area whose intensity value is to be utilized as the initial value for the threshold range. All of the pixels that have the same intensity value will be colored in the image, and displayed in the histogram. The threshold value range initially contains only this one intensity value. As a rule, you will still need to expand this threshold value range. Continue clicking relevant pixels or threshold value ranges, until all of the required structures in the image are a part of the phase.

Click on the *Add Threshold* button, to select additional pixels that are to belong to the threshold value range. The image segments will be colored and displayed in the histogram. The current threshold value range will continue to be expanded until it contains the intensity values of all of the selected pixels.

Click on the *Shrink Threshold* button, to select pixels that aren't to belong to the threshold value range. The threshold value range will continue to be reduced until it no longer contains the pixels you have selected.

Click on the *Undo Pipet* button, to undo the last selections step by step.
Click on the
Redo Pipet button, to restore the last selections that were undone, step by step.

Adding, changing, and deleting phases

Click on the *Add Phase* button, to add a phase for which the threshold values are to be calculated automatically. Double click on the field in the *Phase Name* column, to enter a name for the phase.

Double click on the field in the *Color* column to choose a color. The phase will be displayed in the color you have assigned it, in the image window and in the histogram. The intensity range for the phase will be automatically calculated. In the *Min.[* field, the lower threshold value will be specified. In the *Max.[* field, the higher value will be specified. You can change the values here or you can change them interactively in the histogram.

Click on the *Remove Phase* button, to delete a phase. It's only possible to remove a phase when at least two phases have been defined.

(3) Changing threshold values interactively in the histogram

The histogram shows the intensity distribution of the active image. If the image mainly consists of light and dark areas, the histogram will show two peaks. A peak is an intensity value (or an intensity range) which occurs particularly frequently in the image.

The intensity range which was defined for a phase will be shown as a colored slide in the histogram. You can move the edges of the slide in the histogram. To do that, move the mouse pointer to the edge of the slide. If you have more than one phase, the phase that you would like to change must be selected in the table.

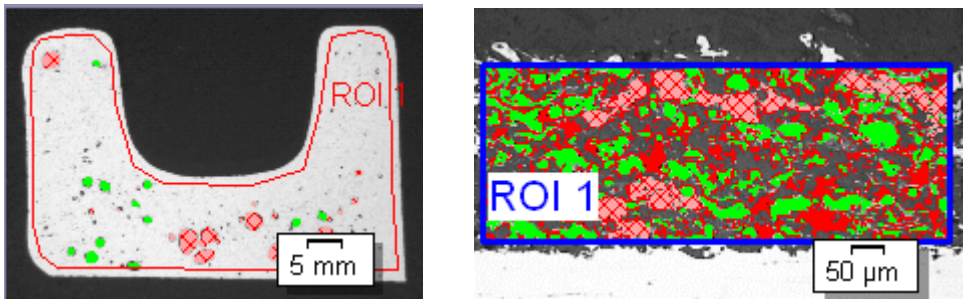
If the mouse pointer changes, click with the left mouse button and drag the edge of the slide in the required direction. The values in the *[Min.* and *Max.[* fields in the table will change. In the image, there will now be more or less pixels in the color of the phase.

10626 04062012

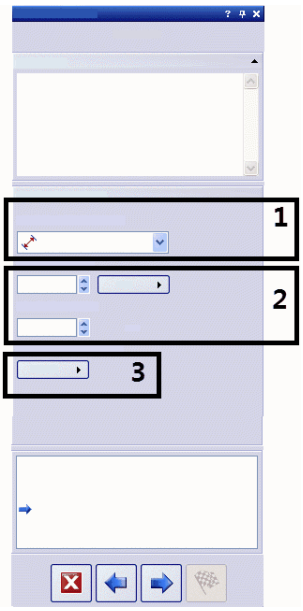
Counting conditions for porosity measurements

All objects which meet the counting conditions, will be shown in this step, in the image, in the color of the phase. If you have defined several phases, the pores will be shown in the color of the phase to which they belong (according to their intensity values).

All objects which do not meet the counting conditions, will be shown in this step with red hatching in the image. This means that these pores (they could be residue particles or artifacts) will not be considered when determining the **pore density** and **pore count**. (They will however still be taken into account when determining the percentage porosity value.)

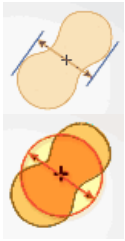


To the left you can see a sample which only has one phase. Pores which do not meet the counting conditions, will be shown with red hatching.
To the right you can see a sample with two phases. Here, pores which do not meet the counting conditions, will be shown with red hatching, too. The red hatching has a fixed specification, so it cannot be seen whether the red hatched objects belong to the green or to the red phase.



(1) "Pore size parameter" field

In this *Pore size parameter* field, you choose how the pore size is calculated.



Choose the *Max. Feret* setting, to use the maximum spacing of parallel tangents on opposing sides of particles.
Choose the *Equivalent Circular Diameter* setting, to use the diameter of a circle which has the same area as the particle.

(2) "Counting minimum" and "Counting maximum" fields

If necessary, click on the button which shows the units (to the right, next to the *Counting minimum* field) and select the units in which the image to be analyzed has been calibrated.

In the *Counting minimum* field, enter the minimum size that an object must have to be considered when determining the number of pores. In the *Counting maximum* field, enter the maximum size that an object may have to be considered when determining the number of pores.

(3) "Unit of pore density" field

If required, in the *Unit of pore density* field, change the units used to show the pore density in the result. The unit is always a unit of area (e.g. 1 mm² or 1 µm²). Your software will then calculate how densely the objects found lie next to one another on the defined surface. The smaller the unit selected, the lower the pore density will be.

10631 04062012

Performing a porosity measurement

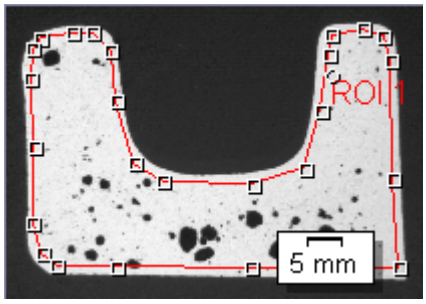
Note: You can follow these step by step instructions on your PC.

Image source step

1. Load the "MacroscopicComponent.tif" example image.
   - The porosity is to be measured in this image.
2. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
3. Click the *Porosity* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
   - By doing so, you skip the *Sample information* step which is not relevant for this example image.
6. Select the *All images* entry in the *Check settings and results* list.
7. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

"ROIs" step

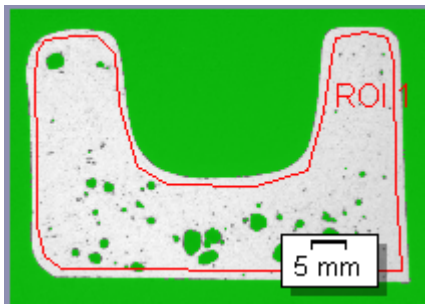
1. For the "MacroscopicComponent.tif" example image, define a polygonal ROI which encompasses the shape of the object. To do that, click the *Create Polygonal ROIs* button, and use several mouse clicks in the image to define the corners of a polygon of arbitrary shape. For the last corner, click using the right instead of the left mouse button.



   - If you would like to measure the porosity on your own images, it may be more sensible to create a circular, triangular or rectangular ROI. You can also define several ROIs with different shapes. The porosity will always be measured over all ROIs.
   - It is not absolutely necessary to define ROIs. If you want to measure the entire image, in the *ROIs* step in the analysis, click directly on the *Next* button without defining a ROI.
2. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

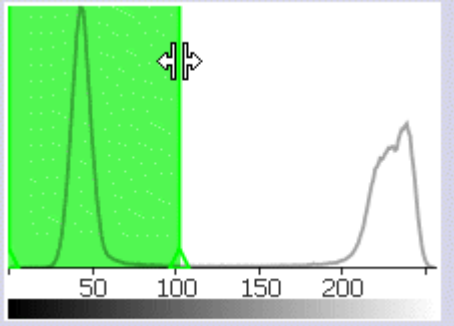
Threshold step

All of the pixels which lie within a defined intensity range, will be shown in color during this step in the analysis. This intensity range is called a "phase". The intensity value range is limited by a top and a bottom intensity value. These are the so-called threshold values. You can find more information on thresholds in the online help.



Please note that the defined ROI will not be considered in this step in the analysis, but only in the next step. This is why the background color in this step is also shown in color.

1. If necessary, reduce or increase the intensity range of the phase. In the image, watch how the object areas found become larger and more objects are found.
   - To do this, change the values in the *Min.* and *Max.* fields in the table in the tool window. Alternatively, interactively change the lower and upper threshold values in the histogram shown at the bottom of the tool window. Move the mouse pointer over the edge of the phase, until the pointer changes and, with the left mouse button pressed, drag the edge in the required direction.



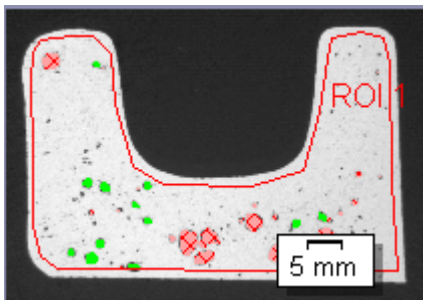
   - One phase is enough for the "MacroscopicComponent.tif" example image. If you would like to measure the porosity on your own images later on, then you may have to define additional phases. You will need two phases for instance, if you would like to measure light and dark pores. You define a second phase by clicking on the *Add Phase* button and also setting the threshold values for the second phase so that the required objects are shown in color.
2. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Counting conditions step

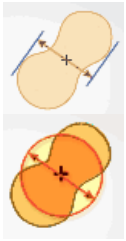
In this step in the analysis, only the parts of the sample within the defined ROIs will be considered.

All objects which meet the counting conditions, will be shown in this step in the color of the phase. If you have defined several phases, then several colors will be used for the display.

All objects which do not meet the counting conditions, will be shown in this step with red hatching. This means that these pores (they could be residue particles or artifacts) will not be considered when determining the **number** of pores found. (They will however still be taken into account when determining the percentage porosity value.)



1. If necessary, you can change the counting conditions. Watch how more or less pores are found as the hatched objects in the image increase or decrease.
2. In the *Pore size parameter* field, choose how the pore size is calculated.



   - Choose the *Max. Feret* setting, to use the maximum spacing of parallel tangents on opposing sides of particles.
   - Choose the *Equivalent Circular Diameter* setting, to use the diameter of a circle which has the same area as the particle.
3. The example image is calibrated in millimeters. You should thus click on the button which shows the units (at the right, next to the *Counting minimum* field), and select mm as the units.
4. In the *Counting minimum* field, enter the minimum size that an object must have to be considered when determining the number of pores. In the *Counting maximum* field, enter the maximum size that an object may have to be considered when determining the number of pores. Watch how more or less objects are found as the hatched objects in the image increase or decrease.

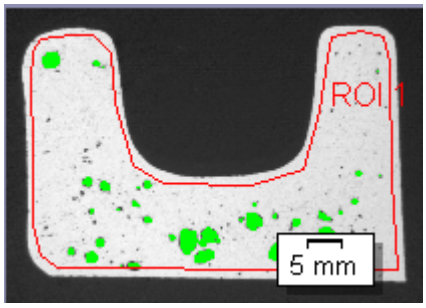
During the analysis process, you can use your software's zoom function as usual. Move your mouse pointer onto the appropriate position in the image, then use the mouse wheel to zoom into, or out of, the image.

5. If required, in the *Unit of pore density* field, change the units used to show the pore density in the result. The unit is always a unit of area (e.g. 1 mm^2 or 1 μm^2). Your software will then calculate how densely the objects found lie next to one another on the defined surface. The smaller the unit selected, the lower the pore density will be.
6. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Image results step

In this step in the analysis, all of the objects used to determine the percentage porosity value will be shown in the color of the phase.

Please note that even objects which were ignored when determining the pore count and were thus shown hatched in the previous step, will be used to determine the percentage porosity value. This is why these previously hatched objects will now again be shown in the color of the phase.



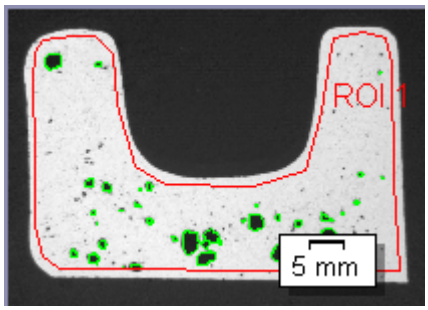
1. Take a look at the results that are shown in the table. In the *Image results* field, you will see the pore count and the percentage porosity value.
2. If necessary, manually change which objects your software uses to determine the percentage porosity value. You can delete, or add objects.
3. Click the *Next* button.

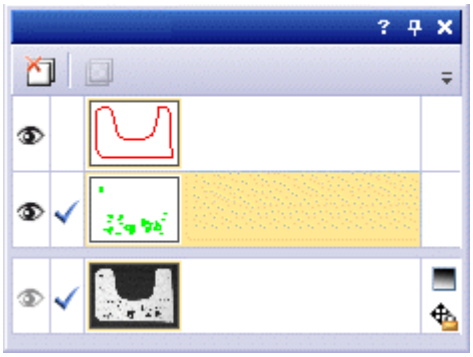
Results step

Select the results you want.

Reporting step

1. Select the *Default* option to use the document template that has been defined as the default document template. The document template determines, e.g. the appearance of the report's header and footer.
2. In the *Content* group, select the check box for the pages the report should contain.
3. Click the *Finish* button.
   - Through the materials analysis measurement, the image has collected one or more additional layers (can be seen in the *Layers* tool window). If required, save the image in TIF or VSI format to retain these newly created image layers.





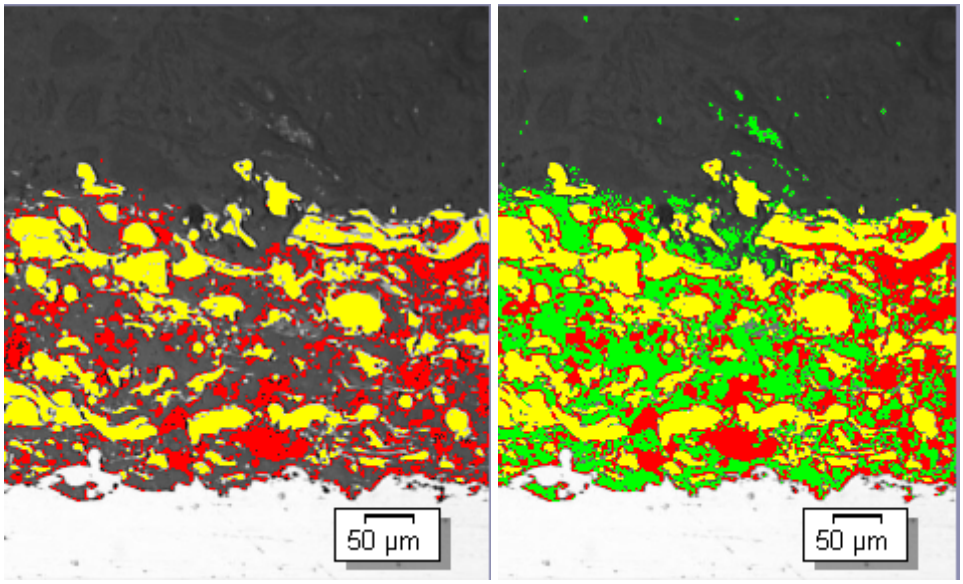
10605 04062012

7.10. Phase analysis

What is a phase analysis?

With a phase analysis, you measure the percentage share by area of the phases in your samples. A phase is a number of pixels which lie within a defined intensity range. The intensity value range is limited by a top and a bottom intensity value. These are the so-called threshold values.

It is a precondition for the phase analysis, that the phases differ from the rest of the sample e.g. because they are darker or lighter. You can define one or more phases. If the parts of the sample (objects), whose percentage area you would like to measure, largely have the same intensity values, then one phase is enough. If the objects have very different intensity values, then several phases will have to be set up.

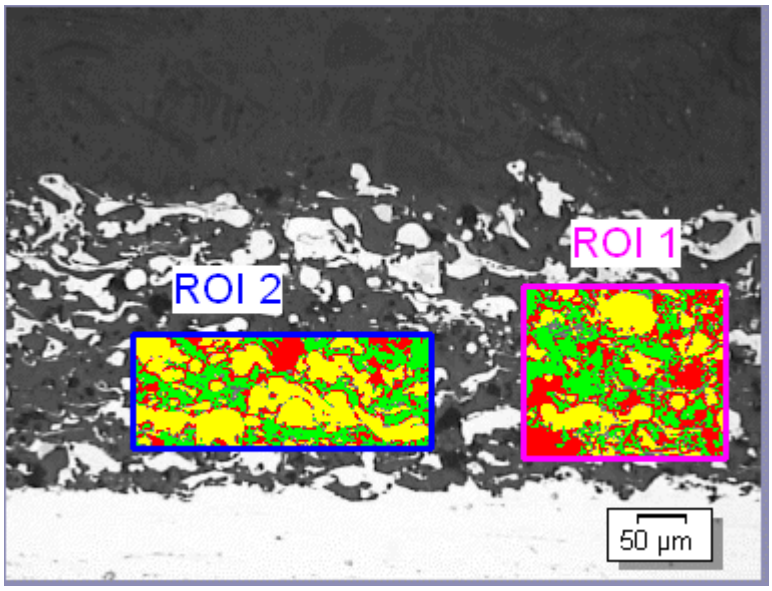


With a phase analysis you can define phases and measure the percentage area covered by this phase. To the left, you can see an example for a phase analysis with two phases (light and dark). In the right example, a third phase was created for the same sample which covers the pixels lying between the dark and the light phase.

The result of the automatic image analysis can be restricted by an object filter. Objects which do not reach the minimum object size will not be considered when determining the percentage area of the phase. In this way you can, for example, prevent dust particles being assigned to a phase and distorting the result.

Measuring on ROIs

You can choose whether you would like to measure the entire image or if the measurement should only be carried out on a part of the image, a so called ROI (Region Of Interest). You can also define several ROIs.



On this image, the percentage area of the phases is measured on two ROIs.

Manually adjusting the result of the automatic image analysis

You can manually adjust the result of the automatic image analysis. You do this interactively on the image. Note that you are not changing the image itself, but the image's measurement layer.

You can manually delete parts of the image which were detected as objects. This can be necessary, if for example artifacts in the image are detected as objects because they have intensity values similar to the defined phase. By manually deleting these objects, the artifacts will no longer be considered when determining the percentage area of this phase.

In addition, you can also manually add other image segments which were not detected as such but which are actually objects. With the manual addition and deletion of objects, you always change the percentage area of the corresponding phase.

Results of a phase analysis

The results of an analysis can be displayed in a workbook. Additionally, or alternatively to that, the results can be displayed in an MS-Word report.

General procedure for a phase analysis

1. Select the analysis process

Click on the *Phase Analysis* button, located in the *Materials Solutions* tool window.

2. "Image source" step

Choose the image you want to measure.

3. "ROIs" step

Define the ROIs, or measure the whole sample.

4. "Threshold" step

Define the intensity range for the first phase, and define other phases if necessary.

5. "Object filter" step

Define the minimum object size.

6. "Image results" step

Take a look at your image results.
If necessary, delete objects, or add new ones.

7. "Results" step

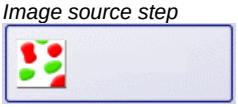
Document the results (report or workbook).

10610 25062012

Performing a phase analysis

Note: You can follow these step by step instructions on your PC.

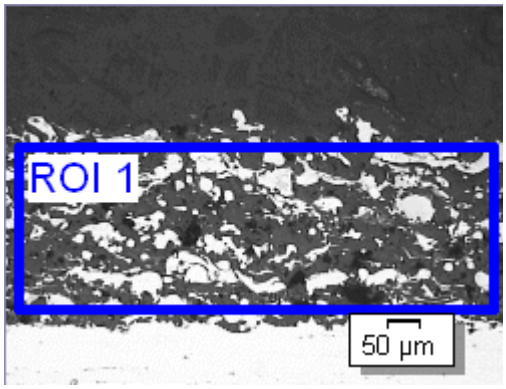
Image source step



1. Load the "SprayCoating.tif" example image.
   - In this image, the percentage area of the bright and the dark phases are to be measured within an ROI.
2. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
3. Click the *Phase Analysis* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
   - By doing so, you skip the *Sample information* step which is not relevant for this example image.
6. Select the *All images* entry in the *Check settings and results* list.
7. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

"ROIs" step

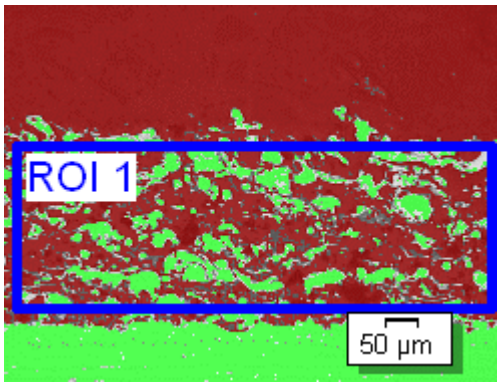
1. For the "SprayCoatingt.tif" example image, define a rectangular ROI, which covers the part of the sample that you would like to analyze. To do so, click the *Create Rectangular ROIs* button, and define the rectangle on the image by two mouse clicks.



   - It is not absolutely necessary to define ROIs. If you want to measure the entire image, in the *ROIs* step in the analysis, click directly on the *Next* button without defining a ROI.
2. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

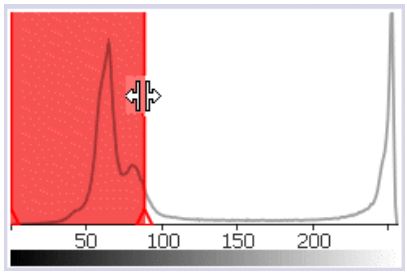
Threshold step

All of the pixels which lie within a defined intensity range, will be shown in color during this step in the analysis. This intensity range is called a "phase". The intensity value range is limited by a top and a bottom intensity value. These are the so-called threshold values. You can find more information on thresholds in the online help.

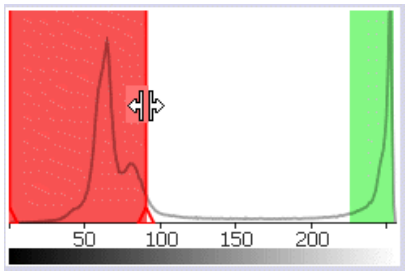


Please note that the defined ROI will not be considered in this step in the analysis, but only in the next step. In this step in the analysis, pixels which are outside of the ROI will thus also be shown in color.

1. If necessary, reduce or increase the intensity range of the first automatically created phase. Make sure that the first phase covers the dark pixels. (You can only define the phase for the bright pixels in the next step.) In the image, watch how the object areas found become larger and more objects are found.
   - To reduce or increase the intensity range, in the tool window's table, change the values in the *Min.* and *Max.* fields. Alternatively, interactively change the lower and upper threshold values in the histogram shown at the bottom of the tool window. Move the mouse pointer over the edge of the phase, until the pointer changes and, with the left mouse button pressed, drag the edge in the required direction.



2. Now define the second phase. To do so, click the *Add Phase* button, and click the *New Threshold* button. Now click so long in the bright areas within the ROI, until they are shown in the color of the phase.
3. If necessary, change the two phases which have already been defined. To do that, select the phase which you want to change in the table in the tool window.

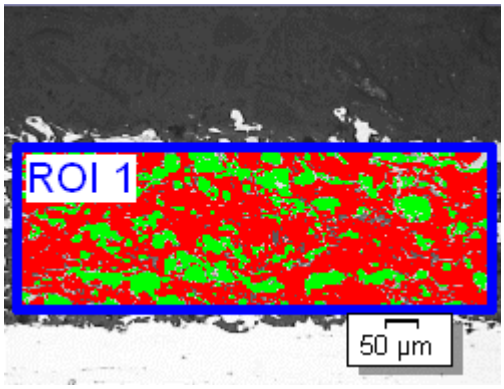


4. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Object filter step

In this step in the analysis, only the pixels within the defined ROI are considered. All objects which meet the conditions defined in the object filter, will be shown in the color of the phases.

All objects which do not meet the conditions defined in the object filter, will be shown in this step in the analysis in red hatching. This means that these objects will not be taken into account when determining the percentage area of the phase.



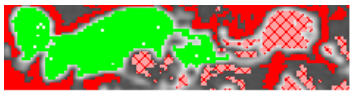
1. If necessary, change the conditions for the object filter. First adjust the units. As the "SprayCoating.tif" image was calibrated in micrometers, click on the button showing the units (to the right next to the *Minimum object area* field), and select µm as unit.
2. In the *Minimum object area* field, enter the minimum size that an object must have to be considered when determining the area fraction covered by the phase. You can thus exclude small objects such as dust particles from the determination of the percentage area of the phase. Watch how more or less object areas are found as the hatched objects in the image increase or decrease.

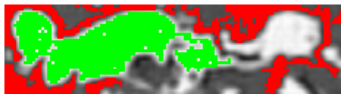
During the analysis process, you can use your software's zoom function as usual. Move your mouse pointer onto the appropriate position in the image, then use the mouse wheel to zoom into, or out of, the image.

3. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Image results step

In this step in the analysis, all of the objects used to determine the phase fraction will be shown in the color of the phase. Objects which do not come up to the minimum area and which were thus shown hatched in the previous step in the analysis, will now be shown with no color.





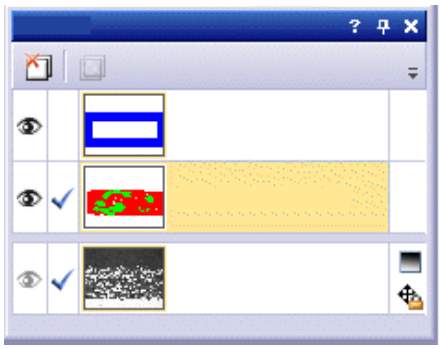
1. Take a look at the results that are shown in the table. In the *Image results* field, you will see the area fraction of each phase.
2. If necessary, manually change which objects your software uses to determine the area fraction of the phase. You can delete, or add objects.
3. Click the *Next* button.

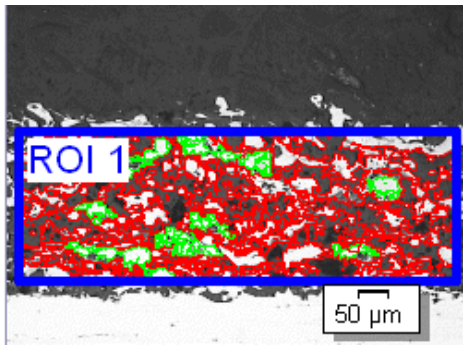
Results step

Select the results you want.

Reporting step

1. Select the *Default* option to use the document template that has been defined as the default document template. The document template determines, e.g. the appearance of the report's header and footer.
2. In the *Content* group, select the check box for the pages the report should contain.
3. Click the *Finish* button.
4. Through the materials analysis measurement, the image has collected one or more additional layers (can be seen in the *Layers* tool window). If required, save the image in TIF or VSI format to retain these newly created image layers.





10611 25062012

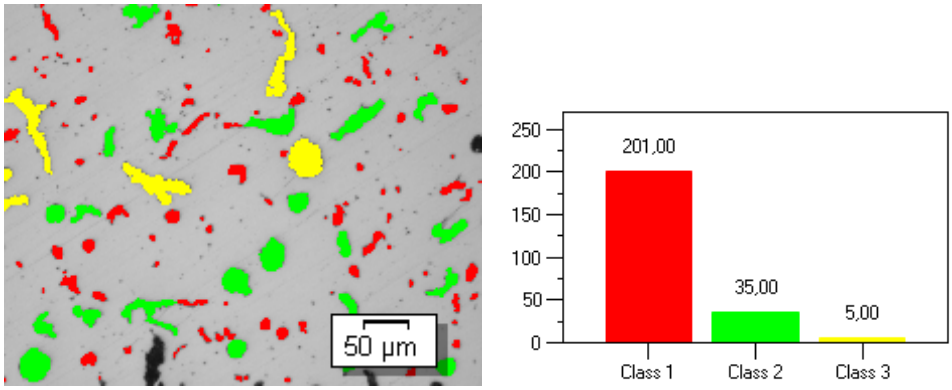
7.11. Particle Distribution

What is a particle distribution?

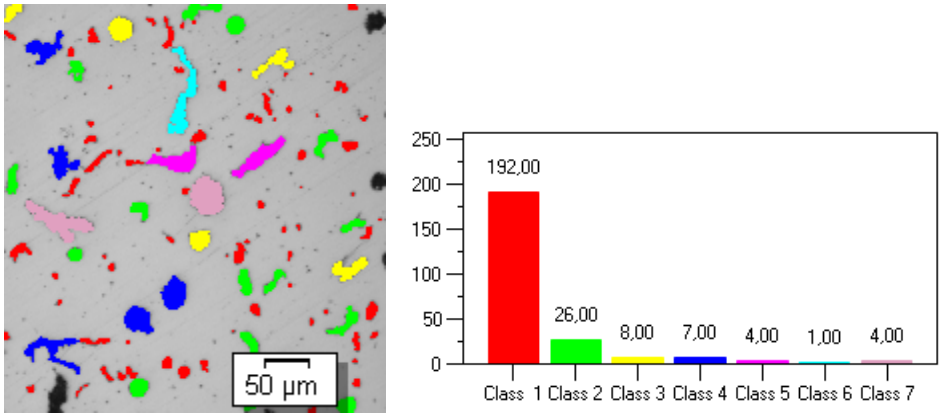
With a particle distribution measurement, the software counts how many particles are in an image and classifies them, for example, according to their size or their form.

It is a precondition that the particles can be detected by the software. Therefore, the particles must differ from the rest of the sample, for example, because they are darker or lighter. In this case, you can define a phase with an intensity range that covers the intensity values of all of the particles. If the particles that you want to measure largely have the same intensity values, then one phase is enough. If you want to measure light and dark particles, you will need a second phase.

All detected particles are measured according to a measurement parameter that you selected (for example, *Area*). The results can be classified automatically. For this, you define a classification with up to 16 classes. For some samples, a coarse classification with only 2 classes is sufficient, whereas other samples require a more detailed classification with 10 classes for example.



Example of a particle distribution measurement. In the image, the particles have been detected and measured according to the *Area* measurement parameter. The results are shown according to the defined classification. In the example shown, the particles were assigned to three size classes. The diagram shows how many particles each size class has.



You see the same particle distribution measurement as in the example above, but now with a more detailed classification. Now the particles were assigned to seven size classes.

Measuring on ROIs

You can choose whether you would like to measure the entire image or if the measurement should only be carried out on a part of the image, a so called ROI

(Region Of Interest). You can also define several ROIs. The particle distribution will always be measured over all ROIs and the results do not differentiate between ROIs.

Manually adjusting the result of the automatic image analysis

You can manually adjust the result of the automatic image analysis. You do this interactively on the image. Note that you are not changing the image itself, but the image's measurement layer.

You can manually delete parts of the image which were detected as particles. This can be necessary, if for example artifacts in the image are recognized as particles because they have intensity values similar to the defined phase. By manually deleting these particles, the artifacts will no longer be considered when measuring the particle distribution. In addition, you can also manually add other image segments which were not detected as such but which are actually particles.

Additionally, you can split particles manually and merge several (previously selected) small particles into one big particle.

Results of a particle distribution measurement

The results of an analysis can be displayed in a workbook and in a chart. Additionally, or alternatively to that, the results can be displayed in an MS-Word report.

General procedure for a particle distribution measurement

1. Select the analysis process

In the *Materials Solutions* tool window, click the *Particle Distribution* button.

▼

2. "Image source" step

Choose the image you want to measure.

▼

3. "ROIs" step

Define the ROIs, or measure the whole sample.

▼

4. "Threshold" step

Define the intensity range of the phase.

▼

5. "Classification" step

Check and change the classification.

▼

6. "Image results" step

Take a look at your image results.
If necessary, delete particles or add new ones. You can also manually split particles and merge separate particles into one.

▼

7. "Results" step

Document the results (report or workbook).

10618 25062012

Measuring the particle distribution

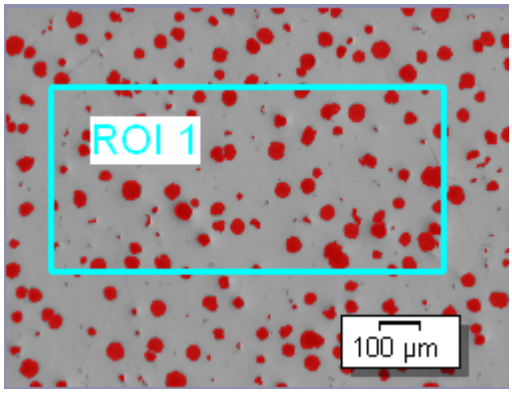
Note: You can follow these step by step instructions on your PC.

Image source step

1. Load the "GlobularGraphite.tif" example image.
   - In this image, the dark nodular graphite particles are to counted and the particles are to be classified according to their size.
2. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
3. Click the *Particle Distribution* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the example image. This image must have been opened for this purpose, and have been selected in the document group.
5. Select the *Skip 'Sample information'* check box.
   - By doing so, you skip the *Sample information* step which is not relevant for this example image.
6. Select the *All images* entry in the *Check settings and results* list.
   - If you analyze your own images later on, you can also select another entry from this list, for example, if you don't want to check the settings for every image anymore.
7. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Threshold step

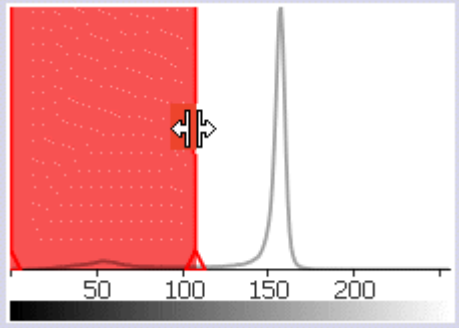
All of the pixels which lie within a defined intensity range, will be shown in color during this step in the analysis. This intensity range is called a "phase". The intensity value range is limited by a top and a bottom intensity value. These are the so-called threshold values. You can find more information on thresholds in the online help.



Please note that the defined ROI will not be considered in this step in the analysis, but only in the next step. In this step in the analysis, pixels which are outside of the ROI will thus also be shown in color.

1. If necessary, reduce or increase the intensity range of the phase. In the image, watch how the particle areas found become larger and more particles are found.
   - To reduce or increase the intensity range, in the tool window's table, change the values in the *Min.* and *Max.* fields. Alternatively, interactively change the lower and upper threshold values in the histogram shown at the bottom of the tool window. Move the mouse pointer over the edge of

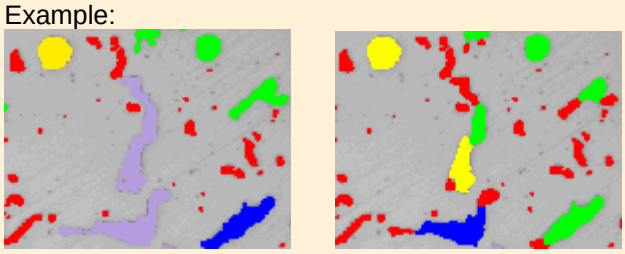
the phase, until the pointer changes and, with the left mouse button pressed, drag the edge in the required direction.



2. For this example, leave the *Auto split particles* check box, found below the histogram, unselected.
   - If this check box is selected, a morphological filter for separating objects is applied before the objects are counted. The parameters, according to which the separation is carried out, are preset.

Note: With the selected "GlobularGraphite.tif" example image, the use of this morphological filter results in only slightly different results. However, with other images, the results may be very different depending on the status of the *Auto split particles* check box. For this reason, check whether for your own images it's better to have this check box selected or not.

Example:



Example of an image for which the status of the *Auto split particles* check box has a significant effect on the image results. On the left is the image where the morphological filter for separating objects has not been applied. Mainly few and large objects have been found. On the right is the same image where the morphological filter for separating objects has been applied. Here, more and smaller objects have been found.

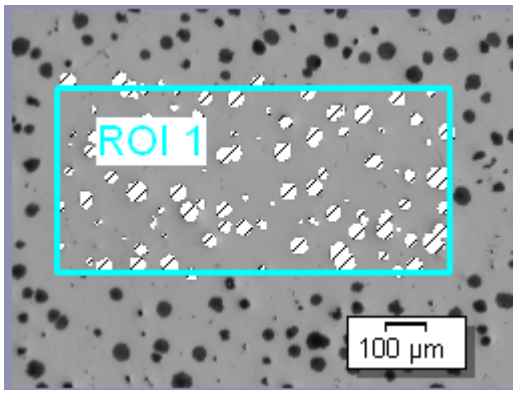
3. Select the *Check classification* check box, which is displayed below the histogram.
   - The additional *Classification* step will be added to the current analysis.

Note: If you analyze several images of a sample in the same analysis process, you can only check the classification for the first image of the sample. The selected classification will be adopted for all other images of this sample. This is why the *Check classification* check box is not available from the second image of the sample onwards.

4. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Classification step

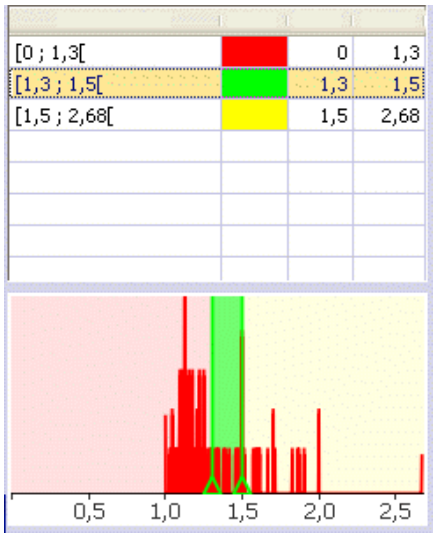
In this step in the analysis, only the pixels within or on the edge of the defined ROI will be considered. All particles that will be used for the particle distribution measurement are shown hatched in this step.



1. As the nodular graphite particles on the "GlobularGraphite.tif" image are to be classified by their size, in the *Measurement* list, select the *Area* parameter.
   - The particle distribution always uses exactly one measurement parameter. The three most frequently used parameters are: *Area*, *Max. (Feret)*, and *Equivalent Circular Diameter*. These parameters are always shown in the *Measurement* list and can be selected quickly.
   - If you analyze your own images later on, you may want to measure the particles according to another parameter, for example, according to the shape. To select another measurement parameter, click the *Select Particle Measurement* button, located at the right hand side of the *Measurement* list. Then select the measurement parameter you want in the *Select Particle Measurement* dialog box. A description of this dialog box can be found in the online help.
2. If necessary, adjust the measurement units. A the "GlobularGraphite.tif"image is calibrated in micrometers, in the *Measurement Unit* field, the*µm²* unit must be selected.

Note: The value in the *Measurement Unit* field depends on the parameter selected in the *Measurement* field. For some parameters, the *Measurement Unit* field is not necessary and will thus not be displayed.

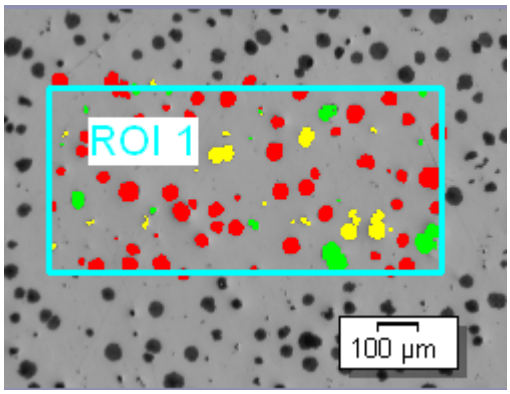
3. Click the *Automatic Classification* button. You will find this button in the toolbar above the table.
   - The *Automatic Classification* dialog box opens.
4. In the *Automatic Classification* dialog box, click the *Get Min./Max. from Image* button. In the *Minimum* and *Maximum* fields, the area of the smallest and biggest particle is entered. The kind of value that is read out from the image and entered in the *Minimum* and *Maximum* fields depends on the selected measurement parameter. In the *Number of classes* field, enter how many classes are to be used for the classification of particles. For the "GlobularGraphite.tif" image, enter the value "3". Close the dialog box with *OK*.
5. Look at the table in the tool window. It contains the classification with the three classes. Also look at the diagram below the table. It displays how many particles are in each class.



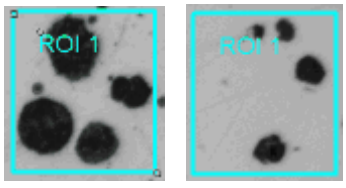
6. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Image results step

In this step in the analysis, all of the particles will be shown in the color of the class to which they belong. All particles that do not belong to any of the defined classes are shown hatched in this step.



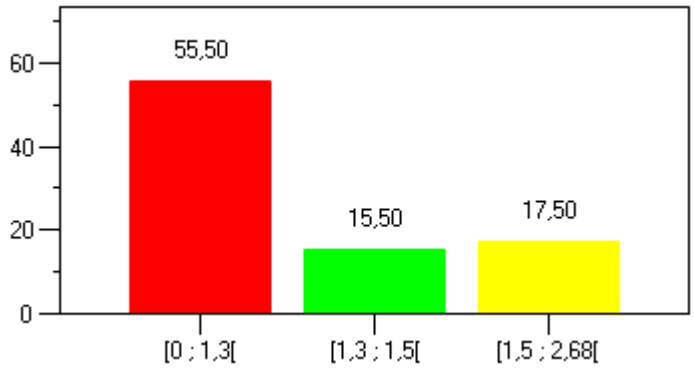
1. Take a look at the displayed results in the *Image results* field. You see how many particles each class contains.
2. The *Particle area fraction* field displays the particle area fraction as a percentage. This value informs you about the percentage the sum of the area of all particles found in this analysis has in comparison to the total area being analyzed (the detection area).
   The particle area fraction is determined by dividing the area of all found particles by the detection area. It doesn't matter whether the found particles have been assigned to a class or not. The detection area can either be the whole image, or one or more ROIs. With particles that are on the border of the detection area, only the part that is inside the detection area is included in the calculation.



On the left is a ROI with a particle area fraction of 40%. On the right is a ROI with a particle area fraction of 10%.

3. In the diagram below the *Image results* field, the classification of the particles is shown graphically. If many classes have been defined, a look a the chart is the quickest way to know which class contains most particles.

Tip: You can also select a different way of classifying the results. Then the chart can look very different. Use the *Tools > Options...* command and select the *Materials Solutions > Particle Distribution* entry in the tree view. This command is not available while an analysis is running.



Note: You will get this diagram as a file in OWB format, if in the *Results* step in the analysis, you select the *Generate chart* check box.

4. If necessary, change the results manually. You can delete and add particles. Additionally, you can split particles manually and merge several (previously selected) small particles into one big particle.
   - Delete particles by first clicking on the particle to be deleted in the image and then clicking on the *Delete selected particles* button. If you respond positively to the warning message, the particle will be deleted. The display in the *Image Results* and *Sample Results* fields is updated. You can also delete several particles at once, by holding the [Ctrl] key pressed, while clicking on the particles.
   - Add particles by first clicking on this button . Then draw a freehand polygon around the particle to be added. Make sure that the freehand polygon lies as exactly as possible on the edge of the particle to be added. End the definition of the polygon with the right mouse button. If you use several phases, from the context menu, select the phase to which the particle added should belong (e.g. *1* or *2*). The polygon added will be shown in the color of the phase. The display in the Image Results and *Sample Results* fields is increased.
   - Merge particles by first clicking in the image on the particles which you want to merge. Keep the [Ctrl] key pressed while you click on the particles. Then click the *Merge selected particles* button.
   - Split particles by first clicking on the *Draw a line that will split particles* button, and then define a line between the particles you want to split. Click the right mouse button and confirm the input. The particle will be split and the split particles will then be shown in the color of the class to which they now belong. Generally this is a different class than before.

Note: If you have manually edited particles and return to the *Threshold* step in the analysis (e.g. to change the threshold values) your manual corrections will be deleted. If necessary, you will then again have to manually edit particles in the *Image results* step in the analysis.

5. Click the *Next* button.

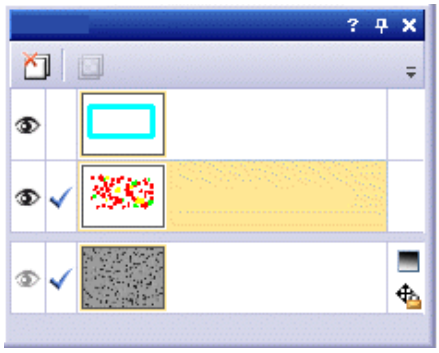
Results step

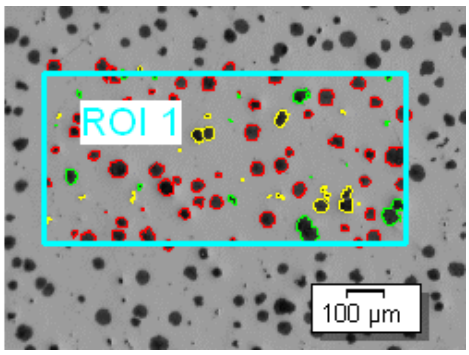
Note: If you analyze several images of a sample in the same analysis process, this step is only shown after the last image has been analyzed.

1. Select the *Generate Report* check box, if you would like to have a report automatically generated in MS-Word once the analysis is completed.
2. Select the *Generate Workbook* check box, to have a document of the "workbook" type automatically created at the end of the analysis.
3. Select the *Generate chart* check box, so that, at the end of the evaluation, the system will automatically create the diagram shown in the *Image results* step as a separate document, of type "workbook".
4. If you want to save the current settings to a file, click the *Save settings....* button. Then assign a descriptive name in the next dialog box.
   - You can load these settings (parameters) when you analyze further images. To do that for the new image in the *Image Source* step, click the *Load from file...* button. The sample and image comments are saved, as are the phases used and the settings in the *Classification* step in the analysis.
5. Click the *Next* button.

Reporting step

1. Select the *Default* option to use the document template that has been defined as the default document template. The document template determines, e.g. the appearance of the report's header and footer.
2. In the *Content* group, select the check box for the pages the report should contain.
3. Click the *Finish* button.
4. Through the materials analysis measurement, the image has collected one or more additional layers (can be seen in the *Layers* tool window). If required, save the image in TIF or VSI format to retain these newly created image layers.





Note: Use the *Tools > Options > Count and Measure > Display* dialog box to specify whether the found particles should be displayed in outline or whether they should be filled. You can change these settings at any time, before or after the analysis, for example, and also for images that have already been saved in TIF or VSI format.

10619 15052013

7.12. Coating Thickness

What is a coating thickness measurement?

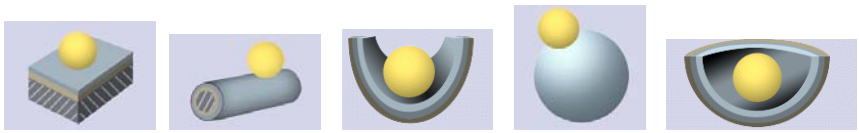
Using the *Coating Thickness* analysis process you can analyze ball indentation cuts of thin coatings and determine their coating thickness. The sample under test should be a substrate, which has one or more coatings that were applied using different coating methods (PVD, CVD, VPS, APS etc.).

To determine the coating thickness, a ball indentation is ground into the sample. This is done using a grinding ball, which has a diameter between about 10 and 50 mm. The ball indentation must have the minimum thickness of the sum of all coatings.

In case of a flat or spherical sample surface, the grinding ball's indentation is round. If the sample surface is curved in one direction, the grinding ball's indentation is ellipse-shaped.

You can choose between the following sample surfaces:

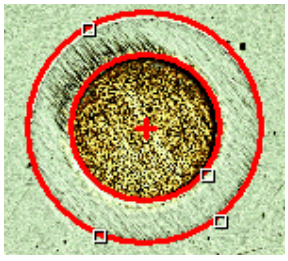
Flat, *Cylindrical convex*, *Cylindrical concave*, *Spherical convex*, or *Spherical concave*.



Measurement order

Coatings are always measured starting on the outside and working towards the inside. This means that, on the image, the coating's outer border is defined first and, following that, the inner border.

The borderlines that have been defined in this way will be shown in color. They are located in an additional image layer (can be seen in the *Layers* tool window). By default, the borderlines are shown in red. You can set a different color or thickness for the borderline.



You can see a coating thickness measurement on the flat sample surface. One coating has been measured.

Results of a coating thickness measurement

The coating thickness is measured in accordance with the industry standard that is set in the program options. The following industry standards are available:

- VDI 3824 : 2001
- EN 1071-2 : 2002

The results of an analysis can be displayed in a workbook. Additionally, or alternatively to that, the results can be displayed in an MS-Word report.

General procedure for a coating thickness measurement

1. Select the analysis process

Click the *Coating Thickness* button, located in the *Materials Solutions* tool window.

▼

2. "Image source" step

Choose the image you want to measure.

▼

3. "Settings" step

Select the sample surface type (*Flat* for example).
Specify further measurement parameters.

▼

4. "Measurement" step

Define the outer and inner borders of each coating.

▼

5. "Results" step

Document the results (report or workbook).

10615 09042013

Measuring the coating thickness

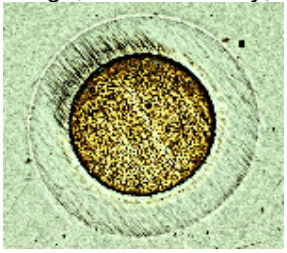
This step-by-step instruction describes how you can measure the thickness of a coating. An image of the flat sample surface on which 1 coating is to be measured has been selected as an example. If you selected an image with a different surface in the *Settings* step, there will be small differences in the procedure.

Example image CoatingThickness2_GrindingBallDiameter_40mm.tif
When your software was installed, several example images were automatically installed at the same time. You can follow this step-by-step instruction using the CoatingThickness2_GrindingBallDiameter_40mm.tif example image. Open this image and make sure that it has been selected in the document group. You can find the information where the example images are located in the online help.

Note: Coatings are always measured starting on the outside and working towards the inside. This means that the coating's outer border is defined first and, following that, the inner border.

Image source step

1. Load the CoatingThickness2_GrindingBallDiameter_40mm.tif example image, or alternatively, the image that you want to measure.



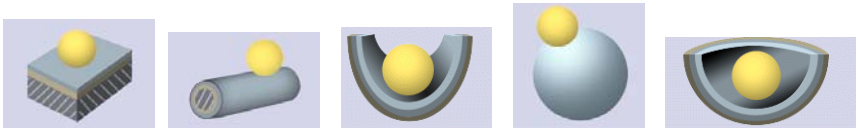
2. Activate the *Materials Solutions* tool window. Should this tool window not be visible, use the *View > Tool Windows > Materials Solutions* command to have it displayed.
3. Click the *Coating Thickness* button.
4. In the *Image source* group, choose the *Selected images* option to analyze the loaded image. For this to work, this image must be selected in the document group.
5. Select the *Skip 'Sample information'* check box to skip the *Sample information* step.
   - As soon as you click on the *Next* button, you'll then go directly to the *Settings* step. You can do this if you don't want to enter any information about the sample, which is the case here.

Note: If you want to analyze images from more than one sample in the same analysis process, the *Skip 'Sample information'* check box must be cleared. Only then will the *New Sample* button be displayed. With this button, you can specify when an image to be analyzed belongs to a new sample.

6. Select the *First image* entry in the *Check settings and results* list.
   - If you select the *First image per sample* entry, you can check the settings for each new sample.
7. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Settings step

1. Select the sample surface type. For the CoatingThickness2_GrindingBallDiameter_40mm.tif example image, select the *Flat* sample surface.
   - You can choose between the following sample surfaces: *Flat*, *Cylindrical convex*, *Cylindrical concave*, *Spherical convex*, or *Spherical concave*.

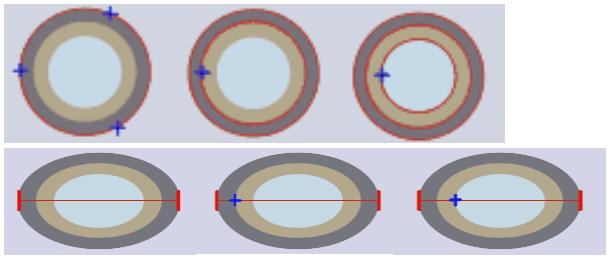


2. Select the crater shape. The indentation that the grinding ball makes in the sample's surface is called a "crater".
   - If the *Flat*, *Spherical convex*, or *Spherical concave* sample surfaces are selected, this indentation is round. The indentation is elliptical when the *Cylindrical convex* or *Cylindrical concave* sample surface types are selected.
3. If you selected the *Cylindrical convex* or *Cylindrical concave* sample surface types: Select the direction of the ellipse's long axis. This information is taken into account when calculating the coating thickness.
4. Specify how many coatings you want to measure in the *Number of coatings* field. A maximum of 20 coatings can be measured. Coatings are always measured starting on the outside and working towards the inside.
5. Enter the diameter of the grinding ball used in the *Grinding ball diameter* field. The grinding ball's diameter must be known in order to produce an accurate coating thickness measurement. If necessary, change the suggested unit.
6. If you selected the *Spherical convex* or *Spherical concave* sample surface types: Enter the curvature radius of the surface used in the *Curvature radius of surface* field. This value must be known because it's needed for the calculation of the coating thickness.

- The curvature radius of the surface is only important for the measurement of the coating of spherical sample surfaces. That's why this field isn't displayed when you've selected a different sample surface type.

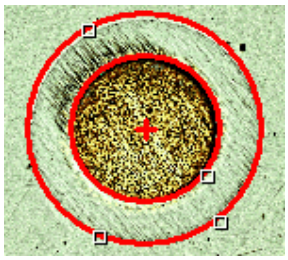
Measurement step

1. Move the mouse pointer onto the image window. All other areas of your software can't be used in this step.
   - The mouse pointer turns into a cross .
2. Define the first coating's outer borderline by clicking three points on its outer border. With cylindrical sample surfaces, the outer borderline is defined by clicking twice on the ellipse's outer border (taking the direction of the selected long axis into account).

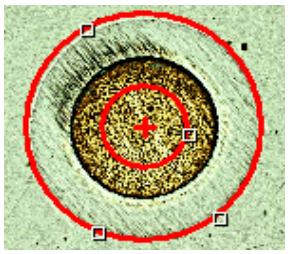


The illustration in the *Measurement* step shows how the borders of a coating have to be defined.

   - The outer borderline is displayed. It's has the color red by default. You can set a different color or thickness for the borderline. Make these settings before you start the analysis process. You can find more information on the analysis process' options in the online help.
3. Define the first coating's inner borderline by clicking once on its inner border.
   - The inner borderline is displayed. If you only want to measure one coating, the mouse pointer turns into an arrow.



4. If you want to measure more than one coating: Define all further coatings to be measured, each with one additional mouse click.
   - As soon as you've defined the inner border of the last coating, the mouse pointer turns into an arrow .
5. Check the values in the *Measurements* table.
6. If you want, you can correct a borderline. To do this, move the mouse pointer to the small handle on the borderline so that it takes on this shape . Now click the left mouse button and move the borderline to where you want it.
   - The borderline is corrected and the values in the *Measurements* table are updated.



7. If you want, you can change the coating's name. Normally, the coatings are numbered serially. If, for example, you prefer to specify the coating material, click once on the number in the *Coating* field in the *Measurements* table to select the entry. Then click on the entry one more time to overwrite it. Enter the text you want.
8. Click the *Next* button.
   - The *Materials Solutions* tool window will display the next step.

Results step

The *Materials Solutions* tool window displays the measurement results.

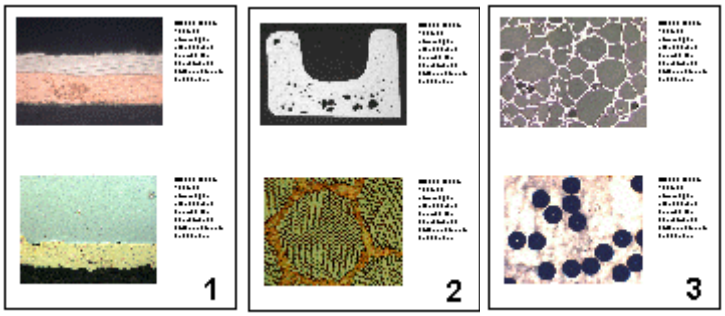
10616 11042013

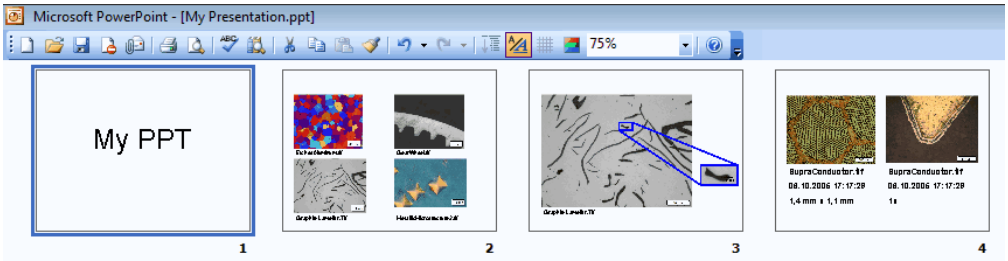
8. Creating reports

8.1. Overview

You can create reports with your software to document the results of your work and to make them available to third parties. You can share reports as files or as printed documents.

Two programs are always involved in the creation of reports: Your image analysis program and Microsoft Corporation's MS-Word or MS-PowerPoint application program.
Therefore, these programs have to be installed on your PC when you work with reports. You can use MS-Word / MS-PowerPoint 2003, 2007 or 2010 for working with reports.





The illustration displays one report in MS-Word format and one in MS-PowerPoint format.

Different ways of generating reports

The requirements for working with reports are very different depending on the user and the way you are working. That's why there are different procedures for creating reports.

Creating MS-Word reports using the "Report Composer" tool window

For users who regularly create reports that are made up in the same way with a lot of images and who require these in MS-Word format.

For this, your image analysis program should be open in the foreground. In the *Report Composer* tool window, open or create a report instruction (RCI file) in which you specify which images and which page layout the report should contain. Then you create a report which is displayed in MS-Word at the touch of a button. In MS-Word you now only undertake small corrections of the report. You can find more information on the *Report Composer* tool window in the online help.

Creating and editing reports using the Olympus MS-Office Add-in

For users who require reports in MS-PowerPoint format. For users that want to insert images or documents that were created with the image analysis program in a new or existing MS-Word document.

For this, your image analysis program should be open in the background. You can use the Olympus MS-Office add-in to insert images, workbooks or charts from your software into an MS-Word or MS-PowerPoint document. You can use templates to do this. With MS-Word reports, you define page templates in the DOC or DOCX file format. With MS-PowerPoint reports, you define slide templates in the PPT or PPTX file format.

The general process flow for report generation using the "Report Composer" tool window

1. New report instruction

Switch to the *Reporting* layout and create or open a report instruction. Place the documents you want in it.

▼

2. Generating a report

Create a report.

▼

3. Working with the Olympus MS-Word add-in

In MS-Word: Have a look at the report and make small changes, if necessary.

▼

4. Saving the report and the report instruction

If you want, print the report or create a PDF file. Save the report instruction (and the report as well, if needed).

The general process flow for report generation using the Olympus MS-Office add-in

1. Creating a new PPT or DOC file

Create a new file or open an existing document.

▼

2. Working with the Olympus MS-Office add-in

In MS-Word or MS-PowerPoint: Use the function in the "Olympus" menu to place documents in the report.

▼

3. Working with the Olympus MS-Office add-in

Use the functions in the "Olympus" menu to define a layout for the report (info stamp, detail zoom and border, for example).

▼

4. Saving the report

Save the report. If you want, print it or create a PDF file.

00112 22042013

8.2. Working with the report composer

The *Report Composer* tool window supports you when you are creating and updating report instructions. In this tool window, you also find the *Create* button that is used to start the report creation.

Note: Two programs are involved in the creation of reports using the *Report Composer* tool window:
Your software and the MS-Word application program. You can use MS-Word 2003, 2007 or 2010 for working with reports.

Should the *Report Composer* tool window be hidden, use the *View > Tool Windows > Report Composer* command to make it appear.

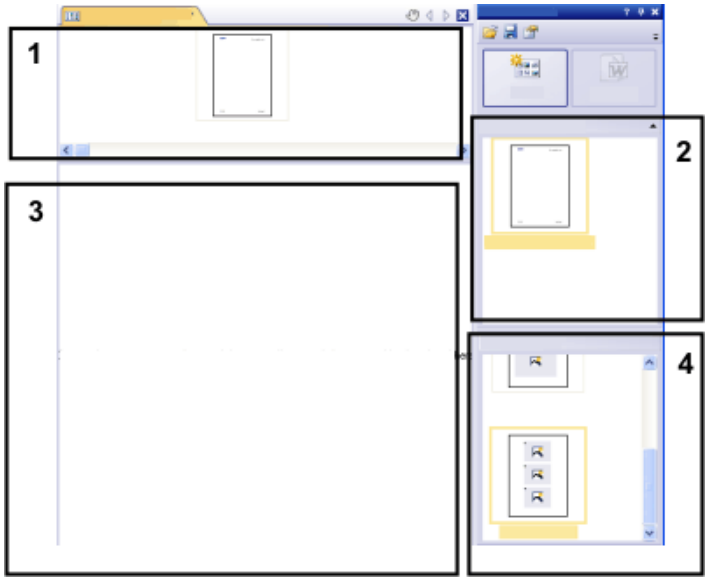
Creating a new report instruction

To create a report, first create a new report instruction in your software. You can also use a saved report instruction.

Note: The report instruction has to contain at least one registered page template. You can find more information on registering page templates in the online help.

1. Switch to the *Reporting* layout.

2. Click the *New Report Instruction* button. You find this button in the *Report Composer* tool window.
   - A new document of the "report instruction" type will be created in the document group. This document is at the same time the workspace in which you put the report together.

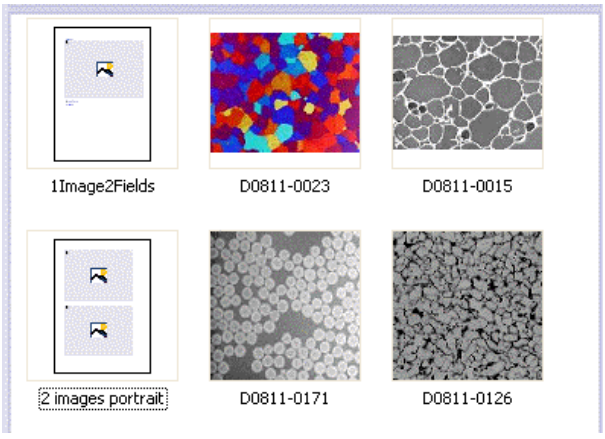


3. If no default document template has been defined: Drag the document template you want onto the upper part (**1**) of the report instruction. You find a list of the available document templates in the upper part (**2**) of the *Report Composer* tool window.
   - If a default document template has been defined, it will be automatically inserted in the upper part of the new report instruction.
   - Creating a report is also possible when you leave the upper part of the report instruction empty. In this case, the default MS-Word document template is used.
4. Drag the page templates you want onto the lower part of the report instruction (**3**). You find a list of the available page templates in the lower part (**4**) of the *Report Composer* tool window.

- Every report has to contain at least one page template.
- Make sure that the page templates contain the correct placeholders for the document types that you want to drag onto the report instruction. Accordingly, if your report is to contain an image and a chart, select a page template that contains one placeholder for an image and another for a chart. You can find more information on page templates in the online help.
- If you want to use workbooks in your reports, MS-Excel must be installed on your PC. The minimum MS-Excel version required is MS-Excel 2003.
- The placeholder for a workbook can also be used for a MS-Excel file. To do so, select the MS-Excel file in the *File Explorer* tool window and drag it onto the report instruction. In the report instruction, MS-Excel files are shown with this icon:

5. Drag the documents you want onto the lower part of the report instruction (**3**).
   - In the *Reporting* layout, the *Database*, *Gallery* , and *File Explorer* tool windows are arranged to the left to the document window. In each of the tool windows you can select one or more documents and drag them onto the report instruction. If you use the *File Explorer* tool window, the documents do not need to be open for this. If you use the *Database* tool window, the documents don't have to be open either. It is sufficient to open the database. However, the *Gallery* tool window only allows you to select documents that are currently open in your software.
   - You can also integrate MS-Word files (e.g., background information regarding the project) into your MS-Word reports. MS-Word file don't need a placeholder in the report instruction. Select the MS-Word file in the *File Explorer* tool window and drag it directly onto the report instruction. In the report instruction, MS-Word files are shown with this icon:

   - The documents must have been saved, because unsaved documents cannot be included in a report.

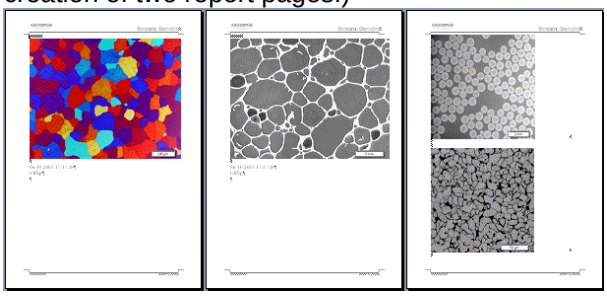


The illustration shows an example of a report instruction. In the report, two different page templates are to be used. The first page template contains a single placeholder for an image, the second page template contains two placeholders for an image. After the page template, the images that are to be inserted in the report page are displayed.

6. Check the report instruction now. You may still edit it and, e.g., delete or shift documents or select another page template.

Creating a report

1. Click the *Create* button. You find this button in the *Report Composer* tool window.
   - The report will be created. Creating a report can take some time when large reports with many images and documents are involved. Pay attention to the progress bar that is shown. The MS-Word application program will open automatically and display the new report. In the example shown below, the report has three pages. (The fact that the first page template only contains one image placeholder and two images have been selected in the report instruction, automatically leads to the creation of two report pages.)



2. If you want to, you can still make additional changes in the MS-Word application program. To do so, use the Olympus add-in or the *Olympus* toolbar.
3. If you want to, save the report instruction and the report.

Editing a report instruction

You can make the changes described below to a report instruction. These changes do not apply to reports the have already been created on the basis of this report instruction. Therefore you must create a new report in order to see the changes you made. This will generate a new MS-Word document. Any changes that you may have made in the first version of the report will not be contained in the newly created DOC-file.

Exchanging the document template

1. Load the report instruction that you want to edit.
   - Report instructions have the file extension RCI.
2. To delete a document template, select it and press the [Del]-key on your keyboard.
3. Drag the new document template onto the upper part of the report instruction.
   - By doing so, the document template is exchanged. Please note that a report instruction can only contain one document template.
   - A report instruction must not contain a document template at all. When you leave the upper part of the report instruction empty, the MS-Word default document template will be taken.

Changing the page templates

1. Load the report instruction that you want to edit.
2. In the report instruction, select the page template you want to exchange.
3. Use the [Del] key on your keyboard to delete the selected page template from the report instruction.

- By doing so, you only deselect the page template, no file will be deleted.

4. Drag the new page template to the position in the report instruction, where the deleted page template had been located.
   - Every report has to contain at least one page template.

Shifting the page templates

1. To shift a page template to another place in the report instruction, select it and, with the left mouse button depressed, drag it to a new position (Drag&Drop).
   - In certain cases, this may change the appearance of the report considerably. All documents that come after this page template in the report instruction will use this page template in the report.

Deleting documents

1. Load the report instruction that you want to edit.
2. In the report instruction, select the documents that you want to delete.
   - The standard MS-Windows conventions apply to the multiple selection.
3. Use the [Del] key on your keyboard to delete all of the selected documents in the report instruction.
   - By doing so, you only undo the document selection, no file will be deleted.

Adding documents

You can add new documents to an existing report instruction at any time.

1. Load the report instruction that you want to edit.
2. Simply drag the new documents onto the position you want in the report instruction.
   - Dragging & dropping images onto the report instruction is possible from the *Database*, *Gallery*, and *File Explorer* tool windows.
   - Please note the page templates must be placed before the images.

Moving documents

You can change the order in which the selected documents are arranged in the report instruction at any time.

1. Load the report instruction that you want to edit.
2. Select an image, and with the left mouse button depressed, drag it to another position (Drag&Drop).

00153 02052013

8.3. Working with the Olympus MS-Office add-in

When your software is installed, an add-in from Olympus is added to the MS-Office MS-Word and MS-PowerPoint application programs. When you start MS-Word or MS-PowerPoint, you can recognize this because the *Olympus* menu is displayed.

Note: Depending on the MS-Office version used, the Olympus add-in looks slightly differently. In MS-Word / MS-PowerPoint 2003, the Olympus menu and the *Olympus* toolbar are available. In MS-Word / MS-PowerPoint 2007 and 2010, only the *Olympus* menu is available. When it is selected, the commands are available in a ribbon.

Note: The language in the *Olympus* menu corresponds to the language set in your image analysis program.

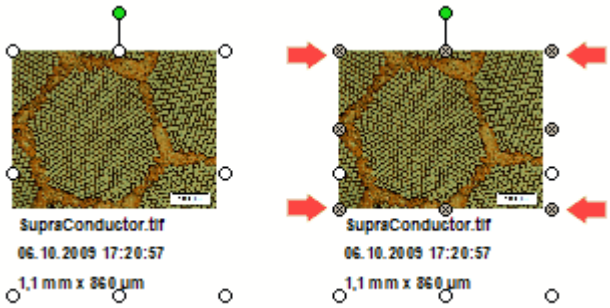
The add-ins' functions

This add-in assists you with very different tasks:

1. Inserting a document that is currently open in your image analysis program, into a MS-Word or MS-PowerPoint document.
2. Inserting a document that is saved locally, or is in your image analysis program's database, into a MS-Word or MS-PowerPoint document.
3. Inserting a field that contains information that is saved in your image analysis program into your MS-Word or MS-PowerPoint document. This makes sense, for example, when you want to see the acquisition date of a certain image.
4. You add one or more detail zooms to an image.
5. You change the image properties and set, for example, whether or not the info stamp and the scale bar should be shown.
6. You change the resolution of one or all images of the report. If you want to share the report, it may be sensible to reduce the resolution, thereby also reducing the file size.
7. You update all placeholders in your report. This makes sense, for example, when you've made changes to the documents in your image analysis program that the report doesn't contain yet.
8. Inserting an MS-Word or MS-Excel document into the software's database. This command is only available if your software supports the database functionality.
9. Defining templates that you want to use for your work with reports. With MS-Word reports, you define page templates in the DOC or DOCX file format. With MS-PowerPoint reports, you define slide templates in the PPT or PPTX file format.

Selecting images in grouped objects

The commands in the *Olympus* menu (*Detail Zoom* for example) can only be carried out when an image has been selected in MS-PowerPoint or MS-Word. You usually select an image with a simple mouse click. When you are working with grouped objects, however, select the whole group with the first mouse click. You can recognize it by the white selection markers. The image only gets selected when you click on it a second time. You can recognize it by the gray selection markers that are displayed additionally.



Left: A grouped object (composed of an image and some text) is selected. Right: The image, a part of the grouped object, is additionally selected. You can recognize it by the gray selection markers (see arrows). Now you can carry out commands in the *Olympus* menu (*Detail Zoom* for example).

Note: During the installation of your image analysis program, some predefined slide templates for working with reports in MS-PowerPoint were installed too. These slide templates contain grouped objects. For this reason, make sure that the image, which is part of the grouped object, is selected additionally before you open the *Olympus* menu.

10404 23042013

Creating and editing a new template

Prerequisite

During the installation of your image analysis program, some predefined templates were installed too. In addition to this, you can define your own templates too. With MS-Word reports, you define page templates in the DOC or DOCX file format. With MS-PowerPoint reports, you define slide templates in the PPT or PPTX file format.

Note: The procedure for creating a template is largely the same, regardless of whether you are creating a page template or a slide template. For this reason, you can carry out the step-by-step instruction with either MS-Word or MS-PowerPoint open.

The contents of a template

In a template, placeholders are set up for the documents that the report is to contain. There are placeholders for images, charts, workbooks and fields. When, for instance, the report is to contain pages that have an image at the top, and below it, a chart, you should then set up a template, which has a placeholder for an image and a placeholder for a chart.

Note: For technical reasons, a template must consist of precisely one page. For this reason, create several separate files if you require several self-defined template pages.

Creating a template and adding a placeholder for a document

1. In the MS-Word or MS-PowerPoint application program, select the *File > New...* command.
2. Select the *Empty Document* option, (MS-Word), or the *Blank presentation* option, (MS-PowerPoint), if you don't want to use an existing template, but instead want to start from scratch.
3. Decide whether you want to insert a placeholder for an image, a chart, or a workbook. Select the *Olympus > Templates* command and choose one of the following options: *Insert Image Placeholder*, *Insert Chart Placeholder*, *Insert Workbook Placeholder*. You can find more information on workbooks in the online help.
   - The placeholder you've selected will be inserted.

4. If necessary, you can change the size of the placeholder. To do so, move your mouse over a handle, then with the left mouse button depressed, drag it in the required direction. The length/width ratio remains unchanged, so that the objects won't be distorted by this action.
5. Doubleclick a placeholder for an image, to change the default settings for its appearance.
   - The *Image properties* dialog box opens. You can find more information in the online help.
6. If required, insert additional placeholders for images, charts or workbooks. Make sure that your template isn't longer than a page.
7. If you want to, you can insert a placeholder for a field. Additional information about a placeholder can be shown in this field, for example, the name, or the date it was set up. You will find additional information on inserting placeholders for fields further down.
8. Save your template. For page templates, use the DOC or DOCX file format. For slide templates, use the PPT or PPTX file format. As a storage location, select the same directory that is set for your user templates or workgroup template in the software.

A note for users creating the MS-Word reports using the Report Composer tool window
If you want you can assign a title to the document. In this case, your software shows this title in the *Report Composer* tool window. If you don't enter a document title, the file name is shown instead in the *Report Composer* tool window. To define a document title, do the following:
In MS-Word 2003: use the *File > Properties* command and switch to the *Summary* tab. Enter the document title in the *Title* field. Close the dialog box with *OK*.
In MS-Word 2007/2010: use the *File > Save as...* command. In the *Save as* dialog box, enter the document title in the *Title* field. Close the dialog box with *OK*.

9. Close the file.

Adjusting the insertion order

The placeholders are numbered in the order in which they were inserted. Should you have initially set up placeholders for two images, have then decided to put a placeholder for a chart right at the top of the page, the insertion order would be that shown in the example on the left.

1. In this case, use the *Olympus > Templates > Adjust Insertion Order* command to have the insertion order numbered serially from top to bottom (see the example on the right).

Inserting a placeholder for a field

1. In the template, select the placeholder into which you want to insert a field.
2. Use the *Olympus > Templates > Insert Field Placeholder...* command.
   - The *Insert Field* dialog box opens. A description of this dialog box can be found in the online help.
   - In the *Placeholder* list, the name of the placeholder into which you want to insert a field appears.
3. In the *Available fields* list, select the field that is to be inserted. The entries in this list are arranged hierarchically. Click the plus sign to expand the list.
   - Two types of field are available.
     The *Document Properties* list contains fields that are, by default, in your software, managed for this document type.
     The *Database fields* list contains all of the fields that are available in the database for the selected placeholder. For this purpose, a database must have been opened.
4. Keep the *Insert Field* dialog box open. Position the mouse pointer on the location in the report where you want to insert the field.
5. In the *Insert Field* dialog box, click the *Insert* button.
   - The placeholder for a field will then be displayed. You can recognize it by the curly bracket, and by the field name shown.
6. If necessary, add placeholders for further fields. To do this, repeat the last 3 steps.
7. Close the *Insert Field* dialog box.
8. Save the template.

00402 25042013

Editing a report

Considerations for users who created the MS-Word report using the "Report Composer" tool window

If you want to make some changes to a report you created using the *Report Composer* tool window, before doing so, you should decide whether it will be better to make the changes in the report (i.e., in MS-Word) or in the report instruction (i.e., in your software).

Often, it is advisable to change the report instruction first and then create a new report. Changes you make in the report instruction are valid for every subsequent report that you create with this report instruction. There are numerous changes that you can anyway, only make in the report instruction, for example, the selection of other page templates.

Changes that you make in a report, are only valid for that particular report. There is also no possible way in which changes that are made in a report can be automatically adopted in the report instruction. However, there are some cases when it makes sense to make a change only in the report, for example when you've created a very large report with a large number of documents, and only want to change the order of two images in it.

If you didn't create the report using a report instruction, but used the Olympus MS-Office add-in in MS-Word or MS-PowerPoint, you will only be able to edit the report anyway.

Note: The following step-by-step instruction relates to reports in MS-PowerPoint and MS-Word formats.

Changing the image properties

When images are transferred to an report, the image link is transferred as well. This makes it possible to change the image display in an report (for example, to scroll the image segment).

1. Double click the image to open the *Image Properties* dialog box. If the image is in a grouped object, first select the group and then double click the image.
2. In the *Display* group, select the *Scale bar if calibrated*, *Info Stamp*, and *Border* check boxes, if these elements are to be displayed.
   - The properties of these elements can be defined in the *Options > Image Information* dialog box. Click the *Options...* button to open this dialog box.
3. In the *Size* group, select one of the options that specify how large the image is to be displayed in the report. You can find more information on these options in the online help.
4. If your settings should apply for all future images, click the *Set as Default* button.
5. Click *OK*.
   - The *Image Properties* dialog box closes. The changed image properties will be shown in the report now.

Adjusting documents

In the report, you can select a document of the "image" or "chart" type and select the *Olympus > Adjust Document* command. You will then change over to the image analysis software, where you can edit the document and then automatically change back to the report.

Example:
In the MS-Word or MS-PowerPoint application program, you edit a report that contains a lot of images. With a certain image you notice that an important measurement is missing. Using the *Adjust Document* command, you change over to the image analysis software, add the missing measurement and then change back to MS-Word or MS-PowerPoint in order to continue editing the report.

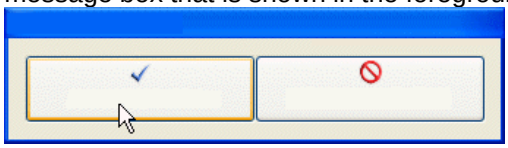
Adjusting an image

1. Open the report and select the image that you want to adjust. If the image is in a grouped object, first select the group and then select the image.
2. Select the *Olympus > Adjust Document* command.
   - You switch to the image analysis software. If it was closed, it will be started and displayed in the foreground.
   - The image that you want to adjust is also opened. In case it is from a database that is currently closed, the database will be opened in the background.

Note: The image analysis software is now in a special "adjust-document" mode. In this mode, you can only make certain adjustments to the image. This is why a lot of other functions are hidden.

3. Make the required change.
4. If the image information was changed: Save the image in the image analysis software.
   - Some changes made to an image don't have to be saved, e.g., when you select another frame in a multi-dimensional image. Other changes have to be saved, e.g., adding a measurement. The fact that a change has to be saved will be indicated by an asterisk shown behind the file name in the document group.

5. Click the *Update Report* button. You find this button in the *Adjust Document* message box that is shown in the foreground.



- MS-Word or MS-PowerPoint will now be shown in the foreground again. The edited image will be displayed. You can now continue to edit the report.
- If your image analysis software was closed before you selected the *Adjust Document* command, it is closed again. If any images or databases had to be opened for this command, they will be closed as well.

Editing a workbook in the report

Your software supports the handling of workbooks. A workbook is created, for example, when you open the *Measurement and ROI* tool window and export a results sheet.

You can find more information on workbooks in the online help.

Note: If you want to use workbooks in your reports, MS-Excel must be installed on your PC. You require MS-Excel 2003, 2007 or 2010.

Apart from the "image" and "chart" document type, reports can also contain workbooks. A workbook is imported as an Excel object in MS-Word or MS-PowerPoint. You can further edit it in the report.

1. In the report, double click on the workbook. If the workbook is in a grouped object, first select the group and then select the workbook.
   - You will change into the edit mode. You can recognize it by the fact that now the column headers and the row numbers are shown. In edit mode, as well as that, you can see all of the workbook's worksheets.
2. If need be, select the worksheet that you want to edit.
3. Double click the workbook in order to switch to edit mode. Make the required change.
   - When you want to format individual cells differently, select the cell and use the *Format Cells...* command in the context menu.
   - When you want to format the complete worksheet differently, (e.g., other font or other background color), select the complete worksheet (e.g., with the keyboard shortcut [Ctrl + A]), then select the *Format Cells...* command in the context menu.
   - When you want to hide a column, click on the column's header, then select the *Hide* command in the context menu.
4. Exit edit mode by clicking on any point in the report, outside the workbook.

Changing image resolution

By default, all images in a report are transferred to reports with a resolution of 192 dpi. In certain cases, it can make sense to change the resolution of individual or all images in a report For example, if you want to print the report, you can raise the resolution. Alternatively, if you want to publish the report on the Internet, you can reduce the resolution.

1. Open the report in MS-Word or MS-PowerPoint. Decide whether you want to change the resolution of all images or just certain images
2. If you only want to change the resolution of one individual image, select that image. If you want to change the resolution of all images, you don't have to select any.

3. Select the *Olympus > Change Image Resolution...* command.
   - The *Change Image Resolution* dialog box opens.
4. Select the option you want in the *Apply to* group. You can choose between *Selected images* and *All images in report*.
   - The *Selected images* option is inactive if no images were selected while opening the command.
5. Specify in the *Image Resolution* group how you want to change the image resolution. If you choose the *User-defined* option, you can enter any resolution of your choice between 96 and 600 dpi into the *DPI* field.
6. Click the *OK* button to change the image resolution.
7. Check whether you are satisfied with the changed image resolution. If not, change the image resolution anew.
   - You can first reduce the image resolution, then save the report and then increase the image resolution again. This is possible because each time that you open the *Change Image Resolution...* command, the image is transferred from your software to MS-Word or MS-PowerPoint.
8. Once you are satisfied with the changes to the image resolution, save the report. Take a look at the new file size in the Windows Explorer.

Updating placeholders

The *Update Placeholders...* command makes it easy to have any changes made to the images after the report has been created also shown in the report. Please take note that all changes made in your software have to be saved, if they are to be displayed when the *Update Placeholders...* command is used.

In MS-Word or MS-PowerPoint, you open a report that you created some time ago. In the meantime, you had changed a lot of images in your image analysis software (e.g., added measurements). Now, the report is to be updated so that it shows the newest version of all of the images.

1. If you only want to update one placeholder, select just that one.
2. Use the *Olympus > Update Placeholders...* menu command.
   - The *Update Placeholders* dialog box opens.
3. In the *Update Placeholders* dialog box, specify whether or not all placeholders should be updated.
4. Select the *Update fields linked with placeholder(s)* check box if your report contains fields which should also be updated.
   - You can find more information on this topic in the online help.
5. Click *OK*.
   - The placeholders will be updated.

Insert Document

You can insert a document at any position in a report. If you have, for example, created a report using the *Report Composer* tool window, and while you are viewing it, notice that you've forgotten an image, you can retroactively insert it into the report.

1. Position the mouse pointer on the location in the report where you want to insert a document.
2. Use the *Olympus > Insert Document...* command.
   - The *Insert Document* dialog box opens.

3. In the area on the left, select the source the document comes from. You have the following possibilities:
   - Select the *Open Documents* entry if you want to insert a document that is currently opened in your software.
   - Select the *Database* entry if you want to insert a document that is part of the currently selected database folder. For this purpose, the database must be opened in your software. Should you work with a version of the software that doesn't support databases, the *Database* entry is hidden.
   - Select the *File Explorer* entry if you want to insert a document that is stored on your PC or in your network.
4. Select the required document in the document preview. Click the *Insert* button.
   - The required document will be inserted into the report.
   - The *Insert Document* dialog box remains open.
5. Insert further documents now or close the dialog box.
   - The path of all documents that you inserted will be saved. That enables you to later update the inserted documents by using the *Olympus > Update Placeholders* command (in case the documents were changed after they have been inserted into the report).

Inserting a field

You can insert a field into a report that describes the image in more detail. All of the values that have been saved in your image analysis program for this image can be displayed in this field.

1. Select the image in the report to which you want to insert a field. If the image is in a grouped object, first select the group and then select the image.
2. Use the *Olympus > Insert Field...* command.
   - The *Insert Field* dialog box opens. A description of this dialog box can be found in the online help.
   - In the *Placeholder* list, the name of the image into which you want to insert a field appears.
3. In the *Available fields* list, select the field that is to be inserted. The entries in this list are arranged hierarchically. Click the plus sign to expand the list.
   - Two types of field are available.
     The *Document Properties* list contains fields that are, by default, in your software, managed for this document type.
     The *Database fields* list contains all of the fields that are available in the database for the selected placeholder. For this purpose, a database must have been opened.
4. Keep the *Insert Field* dialog box open. Position the mouse pointer on the location in the report where you want to insert the field.
5. In the *Insert Field* dialog box, click the *Insert* button.
   - The field contents will be displayed in the report.
6. If necessary, add further fields. To do this, repeat the last 3 steps.
7. Close the *Insert Field* dialog box.
8. Save the report.

Note: Should you want to have the contents of a specific field regularly shown in your reports, you can already insert this field, (that is to say a placeholder for this field) into the page or slide template. Then this field will be automatically filled out

in every report. You can find more information on setting up a template in the online help.

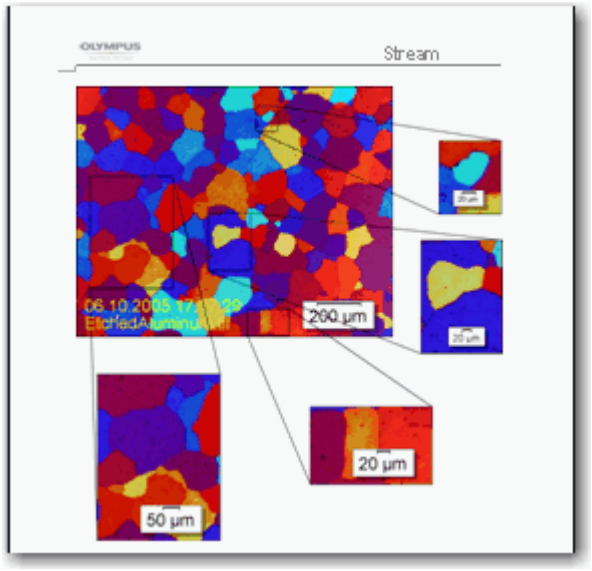
00403 24042013

Working with detail zooms

Note: The following step-by-step instruction relates to reports in MS-PowerPoint and MS-Word formats.

You can insert detail zooms to individual images or all images within a report. By doing so, you can draw attention to especially interesting image details. Each image can have up to eight detail zooms.

The detail zoom factor can be set individually for each detail zoom. The maximum detail zoom factor is 10x. The appearance of the scale bar and the info stamp can be set separately for each detail zoom.

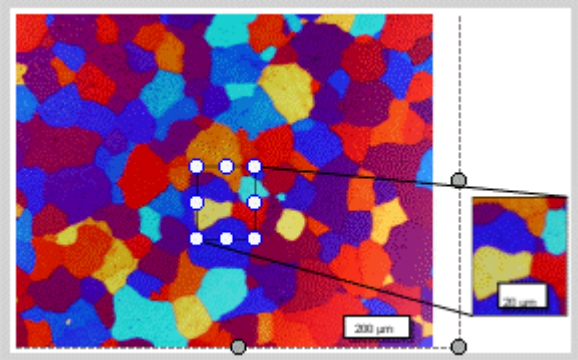


Appearance of an MS-Word report with an image and several detail zooms. In this example, the scale bar is shown in both the image as well as all the detail zooms. Conversely, the info stamp is only shown in the image.

Adding a detail zoom

1. Open the report in MS-Word or MS-PowerPoint and select the image to which you want to add a detail zoom. If the image is in a grouped object, you have to select the group first.
2. Use the *Olympus > Detail Zoom...* command.
   - The *Detail Zoom* dialog box opens. The new detail zoom has already been set up.
3. Select the detail zoom. To do so, click once within the black border in the image.
   - The detail zoom in now selected, which you can tell from the blue handles. All other software controls in the right part of the dialog box will

now become active.



4. Select the detail zoom and drag and drop it to the position you want.
5. Change the detail zoom as necessary (size, shape, magnification factor). You can find more information about this in the next section.
6. If you want to insert more detail zooms: Click the *Add* button and, with the left mouse button pressed, draw a rectangle in the image. Each image can have up to eight detail zooms.
7. Click *OK*.
   - The *Detail Zoom* dialog box is closed. The detail zooms will be shown in the report.

Editing a detail zoom

1. Select the image with the detail zoom that you want to edit. If the image is in a grouped object, you have to select the group first.
2. Use the *Olympus > Detail Zoom...* command.
3. Select the detail zoom that you want to edit. To do so, click once within the black border in the image.
4. Make the required changes. You have the following possibilities:
   - Changing the size and shape
   - Changing the position
   - Changing the detail zoom
   - Changing scale bar and info stamp options
5. If your settings should apply for all future detail zooms, click the *Set as Default* button.
6. Click *OK*.
   - The *Detail Zoom* dialog box is closed. The edited detail zoom will now be shown in the report.

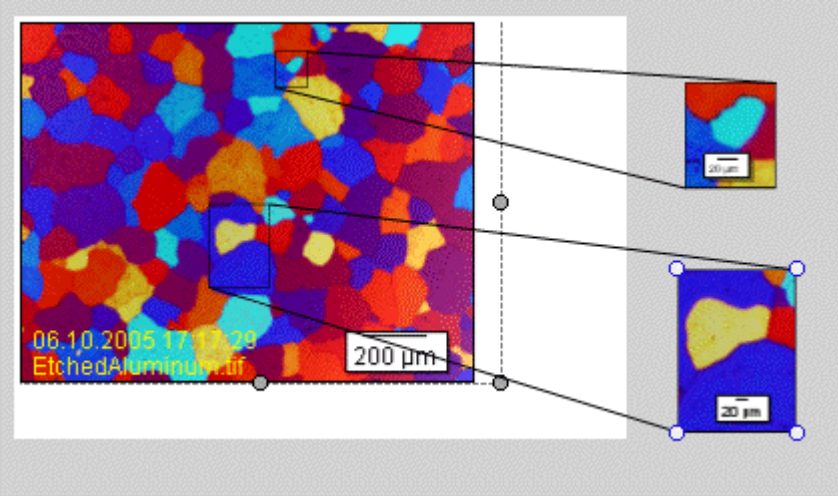
Changing the size and shape

To do so, move the blue handles.

Changing the position

Select the detail zoom and drag and drop it to another position. The detail zooms can be positioned to the right or left of, above or below the image. In certain cases, the images following in the report will be moved onto a new page.

Don't move the detail zoom so far from the image that it no longer fits on the report page.



Part of the *Detail Zoom* dialog box. The report page is indicated by the white background.
In this example, both detail zooms were dragged so far from the image that they no longer fit on the report page. You can see this from the gray background.
In this case, your software shows a corresponding message box. The detail zoom's position can then be adjusted one more time.

Changing the Detail zoom factor

Select the entry you want from the *Detail zoom factor* list. This factor specifies how much the detail will be zoomed. Values from 1x (= not zoomed) to 10x are available. You can also enter a value of your choice into the field and press the [Enter] key.

Changing scale bar and info stamp options

You can specify whether or not the scale bar and info stamp should be shown for each detail zoom separately. Additionally, you can also configure the scale bar's appearance and the contents of the info stamp here. These are the same options that you have for every image in the report (in the *Image Properties> Options* dialog box). As a rule, it is generally a good idea to show the scale in the detail zoom as well.

Deleting a detail zoom

1. Select the image with the detail zoom that you want to delete. If the image is in a grouped object, you have to select the group first.
2. Use the *Olympus > Detail Zoom...* command.
3. Select the detail zoom that you want to delete. To do so, click once within the black border in the image.
4. Click the *Remove* button
5. Click *OK*.
   - The *Detail Zoom* dialog box is closed. The deleted detail zoom will no longer be shown in the report.

00406 25042013

OLYMPUS SOFT IMAGING SOLUTIONS GMBH
Johann-Krane-Weg 39, 48149 Münster, Germany
Phone: +49 (251) 7 98 00-0, Fax: +49 (251) 7 98 00-6060, info.osis@olympus-sis.com